Gebruikershandleiding WAQPAN:
versie Nautilus 1.0 + programma

Project: NAUTILUS

Werkdocument: RIKZ/OS/98.160x

Ministerie van Verkeer en Waterstaat

Directoraat-Generaal Rijkswaterstaat

Gebruikershandleiding WAQPAN:

versie Nautilus 1.0 + programma

Project:	NAUTILUS
Werkdocument:	RIKZ/OS/99.160x
Uitbesteed aan:	ALKYON
Begeleid door:	M. Zijlema

Contents

1 About this manual

This manual is part of the User's guide of WAQUA. For a description of the User's guide of WAQUA, see the section 'General information' of that guide. For a brief description of the program (files, procedure call), the user can refer to the corresponding chapter in the quick reference guide of the User's guide of WAQUA.

Note: this manual differs from the official SIMONA manual and is available only within NAUTILUS group.

The manual of WAQPAN has the following chapters:

- a description of this manual;
- a general description of the program;
- an extensive description of each dialogue in the program;
- a description of each subroutine in the program;
- two appendices;
- glossary;
- index.

In the description of the dialogues of the program, all commands that must be typed by the user will be given in italic characters. All program output will be in a non-proportional font and is indicated by a starting asterisk '*'.

2 General description of the program

2.1 Function

interface

This program provides an interface between the SDS file (WAQUA / TRIWAQ) and the plot packages PRESENT, ANIMATE, MATLAB (including MODELS-GUI) and SIMVIEW (BOX) (if the user wants to select mapdata) or PRESENT and HISPLO (if the user wants to select historydata).

In case of mapdata the user can select data, such as waterlevels, velocities, concentrations and adjoint variables (adjoint waterlevels, U- and V-velocities), at a certain time (momentary data) or data integrated and averaged over a longer time interval (residual data, either Eulerian or Lagrangian quantities). Moreover, the user can also plot the results of the harmonic analysis applied to WAQUA/TRIWAQ runs.

In case of mapdata of a WAQUA- or TRIWAQ-run the user can choose to generate difference maps of two simulations with different times. Furthermore, in case of mapdata of a TRIWAQ-run the user has also the opportunity to generate difference maps of two distinct layers.

There is also a special mode for the MODELS-GUI. With this mode, input files for this GUI are automatically generated.

WAQPAN can be applied after a run of the simulation program. The data files produced by WAQPAN can be used as input to the presentation programs.

WAQPAN reads simulation information written on the SDS file by WAQUA and writes the data to output file(s). Since WAQUA can be executed in WAQUA as well as in TRIWAQ mode, WAQPAN is able to handle both these modes. This is done automatically.

When mapdata of TRIWAQ results are to be generated the user can choose between a horizontal cross section or a vertical cross section. If a horizontal cross section is chosen the user has the possibility to select the depth at which the variables have to be computed. This can be a constant depth with respect to some reference level or a variable depth following the local sigma-layer.
In case of history data either a history plot for a quantity in some special station and in case of TRIWAQ for some special depth or a set of plots over the vertical, for various times for currents and/or constituents in a station, can be selected.

Remarks:
1. In case of Lagrangian quantities depth selection is not available.
2. In case of GUI the handling of horizontal as well as vertical depths and histories is not carried out by WAQPAN but by MODELS-GUI.

output mode

In WAQPAN a parameter has to be specified to indicate which of the possible output modes is desired: PRESENT, ANIMATE, BOX, MATLAB or HISPLO mode. The BOX-mode generates files in the SIMONA-box

structure, which can be included in a SIMONA-input file or can be used within SIMVIEW. In case of MATLAB the user is asked in which format (ASCII or binary) the MATLAB files are generated. In case of MODELS-GUI only MATLAB files in binary format are generated.

Remark : In case of HISPLO an output file is created that should be used by OBSFIL to obtain a correct inputfile for HISPLO. In the following it is assumed that the user is aware of this substep. Both names HISPLO and OBSFIL will be used in the text.

2.2 Output files

subdirectory

WAQPAN generates a subdirectory (in the current directory) containing the requested output file (s).
In the case of PRESENT or OBSFIL mode this directory will contain only one data file with all relevant information of all pictures for the plot program in one sequence. For OBSFIL also a small file with directions for use, as required for this package, is created.
In the case of ANIMATE, MATLAB and BOX mode the directory will contain several data files, for each picture one or more files. If MATLAB is chosen, the user has the possibility to store the data either in ASCII or binary format. If the GUI mode is chosen, the files will always be stored in binary format. Also stored is a metafile which is necessary for the MODELS-GUI. This metafile has the extension **def**. Finally, if requested, a log file will be included in this subdirectory. This file contains relevant information on the data files that have been created.

2.2.1 PRESENT

output file name

For the PRESENT interface only one file is created: PRESENT.OUT.

data

This file contains all selected data. The identification of the data is given by a block code. This is a four-character code generated by WAQPAN. The first block code is 'B001' and this code is augmented by one for each desired output time or each new data selection or each new station. The log file shows the correspondence of block codes, times, stations and data selections.

Some special remarks have to be made with respect to this data file.

mapdata

In case of mapdata the first and second column give the local *x*- and *y*-coordinates. The meaning of the other columns can be found in the header.

positions

Not all output data are positioned at the same grid. In case of horizontal cross sections, depth data are positioned at depth locations and all other output data are positioned at water level locations. In case of vertical cross sections, the *x*-positions are relative distances in the horizontal direction with respect to the beginning of the cross section and the *y*-positions are the values from array ZKS (the centre of the layer), i.e.

positions in the vertical direction.

tidal constants

Concerning the presentation of tidal constants, i.e. astronomical amplitudes and improved kappa-numbers (= local phase lags), of all tidal components that can be found on SDS, these are given for one given (M,N) point. The first column gives the corresponding component numbers, the second column represents the angular velocities and the third and fourth column give the tidal constants.

adjoint variables

In case of adjoint variables, the first and second column give the local *x*- and *y*-coordinates, the third column gives the adjoint waterlevels, while the fourth and fifth column gives the adjoint physical U- and V-velocity components in waterlevel point, respectively.

historydata

In case of historydata the first column always contains the timeseries. The meaning of the next columns can be found in the header. If VERTICALS with historydata are created the actual time, the number, name and position of station, respectively, are found in the header. The first column gives the depth at the position (station) considered.
The meaning of the keywords for data selection can be found in the WAQUA/TRIWAQ section in the SIMONA User's guide.

2.2.2 ANIMATE

For the ANIMATE interface several files are created : mapdata files, boundary outlines files, coordinates files and, if specified by the user, a logfile. In ANIMATE mode no history plots are made.

output file names

ANIMATE requires one data file for each kind of data per output time. These data files are named as follows:

<H/V><block code><selection code><file code><.VEC>.

For example the following list of output files can be generated in one WAQPAN run:

```
H001BDAT            H002LIN             V004G001
H001FDAT.VEC        H003ADAT            V004G002
H001HDAT.VEC        H003CDAT            V004HDAT
H001IDAT            H003EDAT.VEC        V004IDAT
H001LIN             H003G001.VEC        V004LIN
H002ADAT.VEC        H003G002.VEC        V005CCO
H002CDAT.VEC        H003LIN             V005EDAT.VEC
H002EDAT.VEC        V004CCO             V005FDAT.VEC
H002G001.VEC        V004EDAT            V005G001.VEC
H002G002.VEC        V004FDAT            V005LIN
```

whereas in case of tidal analysis, for instance the following output files can be generated:

```
H001LIN
H001MDAT
H001N001
H001N010
H001O025
```

When the SDS-file generated by WAQAD is involved, the following list of output files can be generated in one WAQPAN run:

```
H001CCO
H001LIN
H001PDAT
H001QDAT
H001RDAT
```

All files concerning horizontal cross sections start with an 'H', and all files concerning vertical cross sections with a 'V'.
The block code is a three-character code generated by WAQPAN to identify a certain cross section at a certain time. The first block code is '001' and the code is augmented by one for each desired new output time or each new output window. The log file shows the correspondence of block codes, times and geometries.

The selection code is the code entered by the user to select the kind of data to be outputted. Dependent on the packages WAQUA or TRIWAQ several of the following selections are available :

A[V]	=	Waterlevel
B[V]	=	Bottom Friction
C[V]	=	Depth in meters
E[V]	=	Magnitude of local velocity
F[V]	=	Eddy Viscosity
G<nr>[V]	=	Concentration of Constituent
H[V]	=	Eddy Diffusivity
I[V]	=	Richardson Number
J[V]	=	Eulerian Residual Velocities
K[V]	=	Eulerian Residual Transports
L[V]	=	Lagrangian Displacements

Moreover, if the user wants to plot the tidal constants, instead of the abovementioned list the following selections are given :

M	=	Mean waterlevel or current of tide
N	=	Astronomical amplitude
O	=	Improved kappa-number (local phase lag)

If the user wants to plot the adjoint variables, instead of the abovementioned lists the following selections are given :

P	=	Adjoint waterlevel
Q	=	Adjoint of physical U-velocity in waterlevel-point
R	=	Adjoint of physical V-velocity in waterlevel-point

Remark: In case of residual quantities (J, K and L), tidal constants (M, N and O) and adjoint variables (P, Q and R) only the choice of horizontal cross-sections is supported. Moreover, in case of Lagrangian displacements depth selection is not possible.

The file code can be DAT, LIN, CCO or a three digit long number, if the

selection code is 'G' (constituent) or 'N' (amplitude) or 'O' (phase lag). In the last three cases the number indicates the chosen constituent number and tidal component number, respectively.

An extension 'VEC' is added to the file name when the selection code ended at 'V' (meaning that a velocity field plot is incorporated).

data

For each requested horizontal cross section (in combination with one or more times) only one boundary outlines file (file code = LIN) and only one coordinates file (file code = CCO), in case of a curvilinear model, is created.
For a vertical cross section one boundary outlines file and one coordinates file is created for each output time. In this case the boundary outlines file gives the water level and the coordinates file contains the actual layer thickness for that particular time.

positions

The positions in the coordinates file are:

- for horizontal cross sections:
  (x,y)-positions are the depth locations surrounding the water level locations of the output data (see the user manual of ANIMATE);
- for vertical cross sections:
  the x-positions are relative distances in the horizontal direction with respect to the beginning of the cross section; the y-positions are positions in the vertical direction and are determined with the actual layer thickness.

Note: in case of a vertical cross section the (x,y)-positions in the coordinates file are scaled in order to get a maximum screen filling.

The data in all other files for ANIMATE are positioned at the water level grid.

2.2.3 MATLAB

For the MATLAB interface several files are created : mapdata files, boundary outlines files, coordinates files, files containing the drying/flooding flags and, if specified by the user, a log file. In MATLAB mode no history plots are made.

output file names

MATLAB requires one data file for each kind of data per output time. These data files are named as follows:

<H/V><block code><selection code><file code><.m>.

For example the following list of output files can be generated in one WAQPAN run:

```
H001AMAT.m    H002LIN.m     V004G001.m
H001FMAT.m    H003BMAT.m    V004G002.m
H001HMAT.m    H003G001.m    V004HMAT.m
H001KH.m      H003G002.m    V004KH.m
H001LIN.m     H003IMAT.m    V004LIN.m
H002AVEC.m    H003KH.m      V005CCO.m
H002CMAT.m    H003LIN.m     V005EMAT.m
H002EMAT.m    V004CCO.m     V005FMAT.m
```

```
H002G001VEC.m V004EMAT.m    V005KH.m
H002KH.m      V004FMAT.m    V005LIN.m
```

whereas in case of tidal analysis, for instance the following output files can be generated:

```
H001KH.m
H001LIN.m
H001MMAT.m
H001N017.m
H001O017.m
```

When the SDS-file generated by WAQAD is involved, the following list of output files can be generated in one WAQPAN run:

```
H001CCO.m
H001LIN.m
H001PMAT.m
H001QMAT.m
H001RMAT.m
```

All files concerning horizontal cross sections start with an 'H', and all files concerning vertical cross sections with a 'V'.
The block code is a three-character code generated by WAQPAN to identify a certain cross section at a certain time. The first block code is '001' and the code is augmented by one for each desired new output time or each new output window. The log file shows the correspondence of block codes, times and geometries.

The selection code is the code entered by the user to select the kind of data to be outputted. Dependent on the packages WAQUA or TRIWAQ several of the following selections are available :

A[V]	=	Waterlevel
B[V]	=	Bottom Friction
C[V]	=	Depth in meters
E[V]	=	Magnitude of local velocity
F[V]	=	Eddy Viscosity
G<nr>[V]	=	Concentration of Constituent
H[V]	=	Eddy Diffusivity
I[V]	=	Richardson Number
J[V]	=	Eulerian Residual Velocities
K[V]	=	Eulerian Residual Transports
L[V]	=	Lagrangian Displacements

Moreover, if the user wants to plot the tidal constants, instead of the abovementioned list the following selections are given :

M	=	Mean waterlevel or current of tide
N	=	Astronomical amplitude
O	=	Improved kappa-number (local phase lag)

If the user wants to plot the adjoint variables, instead of the abovementioned lists the following selections are given :

P	=	Adjoint waterlevel
Q	=	Adjoint of physical U-velocity in waterlevel-point
R	=	Adjoint of physical V-velocity in waterlevel-point

Remark: In case of residual quantities (J, K and L), tidal constants (M, N and O) and adjoint variables (P, Q and R) only the choice of horizontal cross-sections is supported. Moreover, in case of Lagrangian displacements depth selection is not possible.

The file code can be MAT, VEC, LIN, CCO, KH or a three digit long number, if the selection code is 'G' (constituent) or 'N' (amplitude) or 'O' (phase lag). In the last three cases the number indicates the chosen constituent number and tidal component number, respectively. The file code 'VEC' is added to the file name when the selection code ended at 'V' (meaning that a velocity field plot is incorporated).

If you have a large amount of data, it may be advised to translate your data into binary format with extension . MAT, instead of ASCII form. This will save your data to the disk. Furthermore, binary MAT-files provide a mechanism for importing your data to MATLAB ‘s workspace in a fast manner. Thus, the following list of output files can be generated in one WAQPAN run:

```
H001CCO.m
H001CCO.mat
H001AMAT.m
H001AMAT.mat
H001EVEC.m
H001EVEC.mat
H001KH.m
H001LIN.m
```

data

For each requested horizontal cross section (in combination with one or more times) only one boundary outlines file (file code = LIN) and only one coordinates file (file code = CCO), in case of a curvilinear model, is created. Also, a file containing the drying/flooding flags (file code = KH) is created.
For a vertical cross section one boundary outlines file and one coordinates file is created for each output time. In this case the boundary outlines file gives the water level and the coordinates file contains the actual layer thickness for that particular time.

positions

The positions in the coordinates file are:

- for horizontal cross sections:
  (x,y)-positions are the depth locations surrounding the water level locations of the output data;
- for vertical cross sections:
  the x-positions are relative distances in the horizontal direction with respect to the beginning of the cross section; the y-positions are positions in the vertical direction and are determined with the actual layer thickness.

Note: in case of a vertical cross section the (x,y)-positions in the coor-

dinates file are scaled in order to get a maximum screen filling.

The data in all other files for MATLAB are positioned at the water level grid.

Example

Let us consider the following files which have been generated during a WAQPAN session:

`H001AVEC.m` contains the water levels (called "Level") and the velocity components (called "U" and "V") of a curvilinear WAQUA simulation.
`H001CCO.m` contains the *x*- and *y*-coordinates of the curvilinear grid (called "Xdep" and "Ydep", respectively).
`H001KH.m` contains the flags for drying/flooding (called "KHU" and "KHV").
`K001LIN.m` contains the boundary outlines (called "Outline").

During a MATLAB session the user can compose a plot containing:

- a checkerboard plot of the water level,
- a vector plot of the velocities,
- the dry cells by lines at the cell boundaries, and
- the boundary outlines.

In order to compose this plot the user has to enter the following commands:

```
>> H001AVEC    % Read the water levels and velocity
>>             % components
>> H001CCO     % Read the (x,y)-coordinates of the depth-
>>             % locations
>> H001KH      % Read the flags arrays for drying/flooding
>> H001LIN                   % Read the boundary outlines
>> pcolor(Xdep,Xdep,Level); % Make a pseudocolor plot
>> colormap(jet);colorbar;  % Set the colormap property and
>>                           % display color scale
>> shading flat; axis equal % Set the shading of plot to
>>                           % flat and make
>>                           % the x- and y-axis equal
>> hold
>> quiver(Xdep,Ydep,U,V)     % Make a vector plot
>> pltkhcc;                  % Plot the 'schotjes'
>>                           % curvilinear case)
>> plot (Outline(:,1), Outline(:,2));  % Plot the boundary
>>                                     % outlines
```

For the plots of the 'schotjes' there are two MATLAB-macro files present, namely PLOTKHU.m (for rectangular models), PLTKHCC.m (for curvilinear models).

In order to plot the curvlinear grid the following MATLAB commands need to be entered:

```
>> c = Level;
>> c(find(abs(c) > 0)) = 0;
>> surface(Xdep,Ydep,c,'Facecolor','w');
>> axis equal;
```

Note that if a binary MAT-file is generated, it is not necessary to load this file with the MATLAB command ‘load’. This will be done automatically, when you just type the name of the corresponding data file.

2.2.4 BOX

For the BOX interface several files are created : data files and (if specified by the user) a log file. All these files concern horizontal mapplots containing actual data only !

output file names

BOX requires one data file for each kind of data per output time. These data files are named as follows:

<H><block code><selection code><file code>.

For example the following list of output files can be generated in one WAQPAN run:

```
H001ABOX          H001G001.BOX
H001BBOX          H001G002.BOX
H002ABOX
H001UBOX
H001VBOX
H001EBOX
```

whereas in case of tidal analysis, for instance the following output files can be generated:

```
H001MBOX
H001N023.BOX
H001O047.BOX
```

When the SDS-file generated by WAQAD is involved, the following list of output files can be generated in one WAQPAN run:

```
H001PBOX
H001QBOX
H001RBOX
```

All files start with an 'H'. The block code is a three-character code generated by WAQPAN to identify a certain cross section at a certain time. The first block code is '001' and the code is augmented by one for each desired new output time or each new output window. The log file shows the correspondence of block codes, times and geometries.

The selection code is the code entered by the user to select the kind of data to be outputted. The following selections are available :

A	=	Waterlevel
B	=	Bottom Friction
C	=	Depth in meters
E	=	Magnitude of local velocity
F	=	Eddy Viscosity
G<nr>	=	Concentration of Constituent
H	=	Eddy Diffusivity

I	=	Richardson Number
J	=	Eulerian Residual Velocities
K	=	Eulerian Residual Transports
L	=	Lagrangian Displacements
U	=	U-component of local velocity at u-velocity point
V	=	V-component of local velocity at v-velocity point

Moreover, if the user wants to plot the tidal constants, instead of the abovementioned list the following selections are given :

M	=	Mean waterlevel or current of tide
N	=	Astronomical amplitude
O	=	Improved kappa-number (local phase lag)

If the user wants to plot the adjoint variables, instead of the abovementioned lists the following selections are given :

P	=	Adjoint waterlevel
Q	=	Adjoint of physical U-velocity in waterlevel-point
R	=	Adjoint of physical V-velocity in waterlevel-point

The file code is BOX or a three digit long number, if the selection code is 'G' (constituent) or 'N' (amplitude) or 'O' (phase lag). In the last three cases the number indicates the chosen constituent number and tidal component number, respectively.

2.2.5 HISPLO

For the HISPLO interface two files are created directly by the program : OBSFIL.OUT and OBSFIL.INP. If the user specified so also a logfile is created. The logfile contains a description of the contents of the file OBSFIL.OUT.

data

The file OBSFIL.OUT contains all selected data. The identification of the data is possible by the layout. The file consists of a number of subparts, the order of which is obligatory : first (if any) the waterlevel part for the selected stations. This block starts with, and this is repeated for each station, a "series" card, (which is a header in which the stationname is given), a "control" card in which the timestep and the number of timesteps are given, next a "format" card in which the format used in the last set of cards is given and finally all waterlevel values. After this "waterlevels block", the "currents block" is found : first for all selected stations the magnitude of the current and next for all selected stations the direction of the current. Next in the same way a block for the U-discharges, a block for the V-discharges and a block for the constituents. In WAQPAN the user is forced to follow the (obligatory) OBSFIL ordering which has been given above. So if, for instance, the currents block is started it is not possible to return to the waterlevels block. The user should keep this in mind while making his selection.
The file OBSFIL.INP is a small file in which some general information as title of the run, date and timestep and stations and quantities are given.

2.2.6 MODELS-GUI

For the GUI interface several MATLAB files in binary format and a metafile with extension **def** are created.

output file names

MODELS-GUI requires one data file per kind of data per output time. The naming of the output files depends on the run-identification of WAQPAN, kind of data, number of layers and number of timestep. The length of the name of the data files is fixed. Hence, if a selection code takes two characters (for example, a constituent called G1), a dummy character is involved for selection code with one character. This dummy character is an underscore. The timestep number corresponds with a real time step which is defined in the metafile. The output files containing the *x*- and *y*-coordinates, waterdepths and waterlevels for each requested timestep are always created. The data files are named as follows:

<run-id><selection code#layer#timestep><.mat>.

Note that the run-identification should not contain the minus-sign. Furthermore, a metafile is created. The name of this file equals the run-identification of WAQPAN, whereas the extension is **def**.

For example, the following list of output files can be generated in one WAQPAN run with identification “g98”:

```
G98A_0001.mat G98G10101.mat G98G10201.mat G98G10301.mat
G98A_0002.mat G98G10102.mat G98G10202.mat G98G10302.mat
G98A_0003.mat G98G10103.mat G98G10203.mat G98G10303.mat
G98A_0004.mat G98G10104.mat G98G10204.mat G98G10304.mat
G98A_0005.mat G98G10105.mat G98G10205.mat G98G10305.mat
G98A_0006.mat G98G10106.mat G98G10206.mat G98G10306.mat
G98A_0007.mat G98G10107.mat G98G10207.mat G98G10307.mat
G98A_0008.mat G98G10108.mat G98G10208.mat G98G10308.mat
G98A_0009.mat G98G10109.mat G98G10209.mat G98G10309.mat
G98A_0010.mat G98G10110.mat G98G10210.mat G98G10310.mat

G98G10401.mat G98G10501.mat G98C_0000.mat
G98G10402.mat G98G10502.mat G98X_0000.mat
G98G10403.mat G98G10503.mat G98Y_0000.mat
G98G10404.mat G98G10504.mat G98.def
G98G10405.mat G98G10505.mat
G98G10406.mat G98G10506.mat
G98G10407.mat G98G10507.mat
G98G10408.mat G98G10508.mat
G98G10409.mat G98G10509.mat
G98G10410.mat G98G10510.mat
```

An example of the corresponding metafile called G98.def can be found in Appendix A.

The selection code is the code entered by the user to select the kind of data to be outputted. The following selections are available :

B	=	Bottom Friction
C	=	Depth in meters
E	=	Magnitude of local velocity
F	=	Eddy Viscosity
G<nr>	=	Concentration of Constituent

H	=	Eddy Diffusivity
I	=	Richardson number
U	=	Physical U-velocity component in waterlevel-point
V	=	Physical V-velocity component in waterlevel-point

If the user wants to plot the adjoint variables, instead of the abovementioned list the following selections are given :

P	=	Adjoint waterlevel
Q	=	Adjoint of physical U-velocity in waterlevel-point
R	=	Adjoint of physical V-velocity in waterlevel-point

data The layer-averaged quantities, except the waterlevel, waterdepth and *x*- and *y*-coordinates, in all layers at all requested time steps are being saved. There is no possibility to make horizontal or vertical cross sections.

positions The *x*- and *y*-coordinates are located in the depth-points, whereas other data are located in the waterlevel-points.

3 User interface

The program WAQPAN is an interactive program, prompting for user selections and producing files to be used by PRESENT, ANIMATE, MATLAB, MODELS-GUI, SIMVIEW or HISPLO.

To see how the program is started, see the quick reference guide of the User's guide of WAQUA.

All prompts of WAQPAN are discussed below together with the possible input and error messages. Lines starting with an asterisk '*' indicate that it concerns a program prompt. Due to the fact that several but different files may be created (map files for fielddata and history files for historydata) the user utilizes only a part of WAQPAN for each specific situation.
The first three prompts are for the identification of the SDS-file, the experiment name and to answer the question whether the user wants map plots (either tidal constants or other data), history plots or plots for MODELS-GUI. These questions are of interest for all users. If the user is interested in mapplots of the following quantities :

- waterlevels
- depth
- velocity components (for BOX U- and V-components can be selected seperately)
- velocity magnitude
- constituent concentrations
- viscosities
- diffusivities
- Eulerian residual velocities
- Eulerian residual transports
- Lagrangian displacements

or if the user is interested in a set of mapplots of

- mean waterlevels or currents of tide
- astronomical amplitudes
- local phase lags

or if the user is interested in a set of mapplots of

- adjoint waterlevels
- adjoint U-velocities
- adjoint V-velocities

then the questions related to the prompts 3.4 up to 3.13 are considered. It should be noted that in the case of tidal constants the prompts 3.7, 3.8, 3.11 and 3.13 are skipped and in case of adjoint variables the prompts 3.4 and 3.7 are skipped.

If the user is interested in mapplots which should be visualised by means of the MODELS-GUI, then the questions related to the prompts 3.24 to 3.28 are considered.

If the user is interested in history plots of the following quantities :

- waterlevels
- currents
- discharges
- constituent concentrations

in some given stations, or if the user is interested in a set of plots over the vertical (TRIWAQ only) of either

- currents or
- constituent concentrations

in one or more stations then prompts 3.14 up to 3.23 are to be given.

The order of the questions while running the program is the same as the order given below, except when explicitly mentioned otherwise. To see quickly how a route through the WAQPAN session is taken, we refer to Appendix B.

It is important to note that there are limitations with respect to the number of constituents the number of stations and the number of times for output which can be treated by WAQPAN. If too many constituents (at this moment 10), or in case of histories too many stations (at this moment 550), or too many times for output (at this moment 9000) are to be considered, the program will give an error message and will stop the execution.

Since WAQPAN may be used to treat different SDS files resulting from both 2D and 3D packages not all possibilities mentioned are available in all cases. In WAQPAN only those prompts that make sense are executed. As an example of this situation: in case of TRIWAQ and a horizontal cross-section the user is asked to select a layer mode, i.e. a horizontal cross-section with variable depth along "sigma layers" or a horizontal cross-section with constant depth. This prompt to determine a layer will never occur if a WAQUA SDS-file is treated simply because in WAQUA no layers exist.

Another important remark concerns the interpolation method that is used whenever the user wants to create a plot of values in a cross-section of constant depth. Due to the definition of the variables in a TRIWAQ computation, some variables like velocities and concentrations are defined in the middle points of the layers. If the value of such a variable is needed in a point situated in either the first half of the first layer or in the second half of the last layer a simple constant extrapo-lation is used in WAQPAN. Interpolation in the other parts is a simple linear inter-polation between the two nearest values.

If points are considered that are permanently dry WAQPAN produces default values to indicate such a situation. In ANIMATE mode a default value 99999. is produced. A value -99999. indicates temporary dry points. In PRESENT mode the value 99999. is used for both situations in

case of scalar quantities. In BOX mode the dry points are given by value 0. In MATLAB mode the dry points are indicated by 'NaN' (Not a Number). In case of vector quantities (like currents) a zero value is produced for all components. Finally in OBSFIL no special "large" defaults are used : if a "dry" station is considered (permanent or temporary) a zero value is submitted both for scalar and vector quantities.

The prompts and their meaning (most prompts are self explanatory):

3.1 Identification of the SDS file

The SDS file you want to work with can be specified here. The program prompts:

```
* Give identification of SDS-file
```

In case of adjoint variables, the SDS-file should have the extension **map**.

If opening of the SDS file failed, the program responds with:

```
* SDS file as proposed does not exist, try again
```

and asks again for an identification.

3.2 Experiment name

The program needs an experiment to read from and prompts:

```
* Give experiment name
```

If opening of the specified experiment failed, the program **stops**.

3.3 Map plots, History plots or plots for MODELS-GUI

The program contains three routes: one for map plots (including tidal constants and adjoint variables), one for history plots and one for MODELS-GUI. The user should indicate which route is wanted.

The program prompts:

```
* Do you want to create MAP files or
* HISTORY files or
* Models-GUI files ?
*
* Type M for MAP files and
* Type H for HISTORY files and
* Type G for Models-GUI files
```

In case of adjoint variables the program prompts:

```
* Do you want to create MAP files or
* Models-GUI files ?
*
* Type M for MAP files and
* Type G for Models-GUI files
```

As already pointed out, the three routes separate here :

If the plotmode is M(ap) then :

First the program prompts for the type of quantities (fielddata) to be plotted

3.4 Type of Mapplots

This question is skipped when adjoint variables are considered.

The program prompts:

```
* Which type of map plot ?
* Make a choice :
*
* type E for Eulerian Residual Velocities
* type L for Lagrangian Res. Velocities
* ( L only allowed for 3D!)
* type T for tidal constants
* type V for other map data.
```

If an error has been encountered in the input the program will give the message:

```
* Only E,L, T or V allowed,try again (--> 3.4)
```

3.5 Output Mode Selection

Next the program asks for the output mode:

```
* To build PRESENT, ANIMATE, BOX or MATLAB files ?

* type 1 or P for PRESENT,
* type 2 or A for ANIMATE,
* type 3 or B for BOX and
* type 4 or M for MATLAB
```

If an error has been encountered in the input the program will give the message:

```
* wrong character found,try again (--> 3.5)
```

If MATLAB mode has been chosen, the following prompt will appear:

```
*  Do you want to store the data in ASCII form
*  or write the data to a binary file ?

*  Type A for ASCII format and
*  type B for binary format
```

Whenever an error is detected in the user input, a message is printed:

```
* Wrong character found,try again
```

3.6 Map Logfile

Next the program will ask whether the user wants to create a logfile:

```
* Give full name of extra log file
* In this file it will be written in
* shorthand what has been sent to the
* outputfile.
*       ( blank = no extra log file )
```

Next the program checks whether the submitted SDS file is a **WAQUA** or **TRIWAQ** file.
In case of residual maps, i.e. Eulerian or Lagrangian quantities, the program checks whether these quantities are available. If not, an error message will appear and the program **stops**.
In case of tidal constants, the program checks whether these constants are available. If not, an error message will appear and the program **stops**.

3.7 Difference maps

This question is skipped in case of tidal constants and adjoint variables. If the requested type of map plot (see Section 3.4) is “other” map data then the program prompts:

```
*  Do you want to make plots
*  of actual map data or
*  of differences of map data

*  type A for actual map data
*  type D for difference map data
```

Whenever an error is detected in the user input, a message is printed:

```
*  Only A or D allowed try again
```

In case of actual map data the program continues with times selection (→ 3.8).
In case of difference map data and in the case that the simulation mode is **WAQUA** the program prompts for the name and experiment of the second SDS-file, whereas if the simulation mode is **TRIWAQ** the following prompt will appear:

```
*  Do you want to have differences
*  of two distinct SDS-files (give S)
*  or of two distinct layers (give L)
```

Whenever an error is detected in the user input, a message is printed:

```
*  Only S or L allowed try again
```

In case of two distinct SDS-files the program ask for the name and experiment of the second SDS-file.

Next, the program prompts:

```
*  Consider two velocity or salinity fields
*  from first and second SDS-files/layers, respectively :
*
*                 u1 and u2
```

```
*
*  The following possibilities are available :
*
```

Only if the simulation mode is **WAQUA** this line is followed by the message:

```
*     absolute : |u1 - u2| (magnitude)
*
*                  u1 - u2   (direction)
```

The next prompt appears in both **WAQUA** and **TRIWAQ** mode:

```
*     nominal : |u2| - |u1| (magnitude)
*
*                  u1          (direction)
*
*     fractional : |u1|/( 0.5*(|u1|+|u2|) )
*
*                  or
*
*                  |u2|/( 0.5*(|u1|+|u2|) ) (magnitude)
*
*                  u1                       (direction)
*
*  type 1 or N for NOMINAL
*  type 2 or F for FRACTIONAL (=u1/(0.5*(u1+u2))
*  type 3 or S for FRACTIONAL (=u2/(0.5*(u1+u2))
```

Note that when the simulation mode is **WAQUA**, the last three lines are replaced by:

```
*  type 1 or A for ABSOLUTE
*  type 2 or N for NOMINAL
*  type 3 or F for FRACTIONAL (=u1/(0.5*(u1+u2))
*  type 4 or S for FRACTIONAL (=u2/(0.5*(u1+u2))
```

The previous prompts will appear only in **ANIMATE**, **BOX** and **MATLAB** mode. If the user specified **PRESENT,** the program prompts:

```
*  Consider two velocity or salinity fields
*  from first and second SDS-files/layers, respectively :
*
*                u1 and u2
*
*  The following possibilities are available :
*
*     nominal : u2 - u1 (componentwise)
*
*     fractional : u1/( 0.5*(u1+u2) )
*
*                  or
*
*                  u2/( 0.5*(u1+u2) ) (componentwise)
*
*  type 1 or N for NOMINAL
*  type 2 or F for FRACTIONAL (=u1/(0.5*(u1+u2))
*  type 3 or S for FRACTIONAL (=u2/(0.5*(u1+u2))
```

Whenever an error is encountered in the user input, the following

message is printed:

```
*  Wrong character found, try again
```

It should be noted that the symbols *u1* and *u2* are treated as velocity vectors or as scalars in case of salinity.

The magnitude of the nominal or fractional difference of velocity and salinity fields will be plotted if the codes ‘E’ and ‘G<nr>’ , respectively, will be chosen in ‘Code Selection’ (see prompt 3.10). The direction of the velocity field will be added to the plots when the selection code ended at ‘V’.

3.8 Map Times Selection

First the time frame, related to the type of plot the user has been asking for, is to be determined. In case of map plot there are two time frames : either an "actual" time frame or a "residual" time frame. As a result of the answer to question given in prompt 3.4 the program knows which frame has to be selected. Note that in case of tidal constants this prompt will be skipped. Next the program prompts for the selection of times (this selection is to be done by the user).

The program prompts :

```
*   Title of the SDS-file involved:
*    ....................................................
*
*
*   Time frame SDS file : File Time First   =   ........
*                         File Time Interval=   ........
*                         File Time Last    =   ........
*
*     The following possibilities are available :
*
*        a single time : time
*
*        several times (1 to N) : time1,..,timeN
*        (separate two values with a comma)
*
*        if you want all times available then give: *
*
*        if you want a new time interval then give either I (time)
*        or i (time)
*        with (time) the new interval. New multiple from ......
*        will be determined by the program.
*
*     Separate times with comma's, close your input with a "/"
```

It is allowed to enter more than one line of input. The input will be closed when a slash is given on the end of a line. Per input line you can give :
- one time : this time will be added to the list of selected times
- several times : these times will be added to the list of selected times
- an asterisk : all available times will be added to the list of selected times

- interval i<dt> or I<dt> : this has the same effect as giving an asterisk, only the new given time interval dt will be used. Note that no spaces in between **i** and **dt** are allowed

Every time that an input line is given, the program shows the selected time(s) i.e. one or more times the following line if you chose one of the first two possibilities :

```
Added <time>
```

and the lines :

```
times added with interval <time interval>
starting with <time first>
ending with <time last>
```

if you chose the third possibility.

In the program the next checks are done and (if needed) adaptions are made during the analysis of your input :

- A single time must fit within the time range. When it does, the nearest time available is added.

- A new time interval is rounded to a whole number of time steps.

Whenever an error is detected, a message is printed and the rest of the input line is analyzed further.

If difference map of two different SDS-files is required, the program prompts after the Map Times selection of the first SDS-file:

```
* Times for difference maps
*
* You have chosen to make difference maps from different
* SDS-files.
* Do you want to use the same times as for the first
* SDS-file (Y/N)
```

If the user chooses to use different times, the program prompts for the 'Map Times Selection' (→ 3.8) of the second SDS-file/experiment.

3.9 Geometry window

The program prompts for the geometry window of the map to be plotted :

```
* Give the coordinates of the cross-sectional window:

 * MLEFT,  MRIGHT,  NDOWN,  NUPPER
```

only if the simulation mode is **TRIWAQ** this line is followed by the message:

```
* (for a vertical cross-section either M or N should be
   constant)
```

The input given by the user is checked :

- Whenever an M or N input value is out of range it is replaced by the minimum value available (for MLEFT and NDOWN) or the maximum value available (for MRIGHT and NUPPER).
- Moreover the following inequalities should hold :
  MLEFT <= MRIGHT and NDOWN <= NUPPER.

Any error results in a message after which the geometry line may be entered again.

In case of the presentation of tidal constants in **PRESENT** mode, the user should give one point only, i.e. MLEFT = MRIGHT and NDOWN = NUPPER. In that case, the program prompts:

```
* In case of presentation of tidal constants in
* PRESENT mode, give the coordinates of one point :

*   M, N
```

In case of a vertical cross-section only one active line is allowed. This means that if the cross-section line starts in an inactive point, crosses the water to an island and goes on over the water to another island, only the first part of water, i.e. the first set of active points, is treated in **PRESENT**, **ANIMATE** and **MATLAB** mode.
In case of residuals or tidal constants or adjoint variables no vertical cross-sections are allowed. If the user, in spite of this restriction, asked for a vertical cross-section, he will be invited to try again !
In case of difference map of two distinct layers no vertical cross-sections are allowed. If the user, in spite of this restriction, asked for a vertical cross-section, he will be invited to try again !

3.10 Code Selection

In case of a residual computation due to the answers already given the output is (almost) completely determined :

if Eulerian quantities have been selected in question given in prompt 3.4 then an output file with coordinates and either at the users wish Eulerian residual velocities (i.e. the integrated velocities divided by the total time-interval considered) or Eulerian residual transports (i.e. the, over the time interval considered, integrated transports divided by the, integrated, layer thicknesses) will be printed. The screen message reads :

```
* Give indication for the MAP(s) wanted:
*  1 :  u,v as vector with their magnitude
*  2 :  Eulerian transports with their magnitude
```

If an error is found (user did not submit 1 or 2) the message repeated and the user should try again !

else if Lagrangian quantities have been selected in question given in prompt 3.4 then an output file with coordinates and Lagrangian

"velocities" (i.e. Lagrangian displace-ments divided by the length of the time interval considered) will be created. In this case no input is needed since only one possibility is available.

In the case of other quantities, except the tidal constants and the adjoint variables, the user has to enter a (or several) code (s) to select the type of data to be outputted. All input must be given on one line, with the choices seperated by comma's only. An input error results in a message after which the complete line must be entered again. The following possibilities are available :

- if the output mode is **PRESENT** and the simulation mode is **WAQUA** next prompt : 10A
- if the output mode is **ANIMATE** or **MATLAB** and the simulation mode is **WAQUA** next prompt : 10B
- if the output mode is **PRESENT** and the simulation mode is **TRIWAQ** and the cross-section mode is HORIZONTAL next prompt: 10C
- if the output mode is **PRESENT** and the simulation mode is **TRIWAQ** and the cross-section mode is VERTICAL next prompt: 10D
- if the output mode is **ANIMATE** or **MATLAB** and the simulation mode is **TRIWAQ** and the cross-section mode is HORIZONTAL next prompt : 10E
- if the output mode is **ANIMATE** or **MATLAB** and the simulation mode is **TRIWAQ** and the cross-section mode is VERTICAL next prompt : 10F
- if the output mode is **BOX** and the simulation mode is **WAQUA** next prompt : 10G
- if the output mode is **BOX** and the simulation mode is **TRIWAQ** and the cross-section mode is HORIZONTAL next prompt : 10H

```
10A      PRESENT / WAQUA

     *   Give codes for horizontal cross sections:
     *     SE = Waterlevel
     *     H  = Depth in meters
     *     VC = Vel.Components + Constituents

10B      ANIMATE-MATLAB / WAQUA

     *   Enter codes, a "V" added to a code means
     *   that a vector plot will be incorporated :
     *   A[V]      = Waterlevels
     *   C[V]      = Depth in meters
     *   E[V]      = Magnitude of local Velocity
     *   G<nr>[V] = Concentration of Constituent number<nr>

10C      PRESENT / TRIWAQ / HORIZONTAL CROSS-SECTION

     *   Give codes for horizontal cross sections:
     *     SE = Waterl. + Bottom stress vel. + Chezy
     *     H  = Depth in meters
     *     VC = Vel.Comp. + Visc + Diff + Rich + Const
```

```
10D      PRESENT / TRIWAQ / VERTICAL CROSS-SECTION

     *   Give codes for vertical cross sections:
     *     SD = Waterl. + Disch + Bottom stress + Chezy
     *     VC = Vel.Vectors + Visc + Diff + Rich + Const

10E      ANIMATE-MATLAB / TRIWAQ / HORIZONTAL CROSS-SECTION

     *   Enter codes for horizontal cross sections
     *   A "V" added to a code means that a vector
     *   plot will be incorporated :
     *   A[V]     = Waterlevels
     *   B[V]     = Bottom Friction
     *   C[V]     = Depth in meters
     *   E[V]     = Magnitude of local Velocity
     *   F[V]     = Eddy Viscosity
     *   G<nr>[V] = Concentration of Constituent number<nr>
     *   H[V]     = Eddy Diffusivity
     *   I[V]     = Richardson numbers

10F      ANIMATE-MATLAB / TRIWAQ / VERTICAL CROSS-SECTION

     *   Enter codes for vertical cross sections
     *   A "V" added to a code means that a velocity
     *   field plot will be added to the plot:
     *   E[V]     = Magnitude of local Velocity
     *   F[V]     = Eddy Viscosity
     *   G<nr>[V] =Concentration of Constituents number<nr>
     *   H[V]     = Eddy Diffusivity
     *   I[V]     = Richardson numbers

10G      BOX / WAQUA

     *   Enter codes :
     *   A     = Waterlevels
     *   C     = Depth in meters
     *   U     = Local Velocity in U-direction
     *   V     = Local Velocity in V-direction
     *   G<nr> = Concentration of Constituents number <nr>



10H      BOX / TRIWAQ / HORIZONTAL CROSS-SECTION

     *   Enter codes for horizontal cross sections :
     *     A     = Waterlevels
     *     B     = Bottom Friction
     *     C     = Depth in meters
     *     U     = Local U-velocity in U-velocity point
     *     V     = Local V-velocity in V-velocity point
     *     E     = Magnitude of local Velocity
     *     F     = Eddy Viscosity
     *     G<nr> = Concentration of Constituent number <nr>
     *     H     = Eddy Diffusivity
     *     I     = Richardson numbers
```

If harmonic analysis of tides has been selected (see prompt 3.4) the user has to enter one or more codes to select the type of tidal constants to be plotted in **ANIMATE**, **BOX** or **MATLAB** mode. If the output mode is **PRESENT**, this prompt will be skipped. All input must be given on one line, with the choices seperated by comma's only. An input error results in a message after which the complete line must be entered again. The

following possibilities are available :

```
*   Enter codes :
*     M     = Mean waterlevel or current of tide
*     N     = Astronomical amplitude
*     O     = Improved kappa-number (=local phase lag)
```

The user should make a selection of tidal components for which the astronomical amplitude and local phase lag should be plotted. To this end the program gives the next prompt:

```
* Number of tidal components is ...
* Make selection for astronomical amplitude/local phaselag
* Do you want all tidal components on screen ?
* Give y(es) or n(o)
```

If the screenmode is y(es) this will result in a list of names of tidal components of the form:

```
* Available components are

*  ... ... ... ... ... ... ... ...
*  ... ... ... ... ... ... ... ...

 etc.
```

Next, the names of the tidal components have to be entered:

```
*  Give sequence names of components
*  that you want to select and
*  separate names with comma's
```

If an error has been encountered in the input an error message will appear and the user should try again.

In case you have chosen adjoint variables, you have to enter one or more codes to select the type of adjoint variable to be plotted in **ANIMATE**, **BOX** or **MATLAB** mode. If the output mode is **PRESENT**, this prompt will be skipped. All input must be given on one line, with the choices separated by comma's only. The following possibilities are available:

```
*   Enter codes :
*     P     = Adjoint waterlevels
*     Q     = Adjoint U-velocity component in wl point
*     R     = Adjoint V-velocity component in wl point
```

If an error has been encountered in the input an error message will appear and the user should try again.

3.11 Depth Determination

In the case that the simulation mode is **TRIWAQ** and the cross-section mode is HORIZONTAL (i.e. N nor M given in the geometry card are constant) the user may enter a code to select a layer mode , i.e. a horizontal cross-section with variable depth or a horizontal cross-section with "constant depth". This prompt will be skipped when the tidal constants have been chosen. In case of adjoint variables only layer mode is automatically selected. When the choice "constant depth" has been made the user has to indicate a depth mode , i.e. constant depth with respect to the waterlevel or constant depth with respect to the reference level of the simulation. If difference map of two distinct layers is requested, only a horizontal cross-section with variable depth is allowed.

Unless, the difference map of two distinct layers is requested, the program prompts:

```
* Do you want horizontal cross-sections along
* lines of variable depth (i.e. sigma layers )
* (give s) or horizontal cross-sections
* with constant depth (give d)
```

If an error is detected in the layer mode, the program will continue :

```
* Only s/S/d/D allowed, try again ! ( --> 3.11)
```

Next, if layer mode is s/S (sigma) or difference map of two different layers is requested :

```
* Give number of sigma layer to be considered
* Only positive integers allowed
* between 1 and <kmax>
```

If an error is found in the layer number the program will respond :

```
* Only positive integers allowed
* between 1 and <kmax>
```

If layer number is ok next prompt is --> 3.12 or, if difference map of two distinct layers is requested, the following prompt will appear :

```
* You have chosen to make difference
* maps from different layers.
*
* Please enter the required layer number
* for the second map.
*
* Only positive integers allowed
* between 1 and <kmax>
```

else if layer mode is d/D :

```
* Horizontal cross-sections of constant depth
* with respect to waterlevel (then give w) or
* with respect to reference level (give r)
```

If an error is encountered in the depth mode :

```
* Only w/W/r/R allowed, try again !
```

If depth mode is accepted then :

```
* Give depth to be considered
* Downwards is positive !
```

If depth mode is ok next prompt will be → 3.12

3.12 Choice to Continue

After the creation of these map files the program can go on to create more map plots. The user has the possibility to force the program to do so answering the following question :

```
* Do you want to continue this WAQPAN session ?
* Give y or n
```

If answer is n(o) ----> END OF SESSION. Program stops.
If answer is y(es) ----> 3.13

3.13 Same times Frame ?

If the answer for continuation was yes then the program starts again but first it is checked whether a time selection has to be done again or not (except for tidal constants) :

```
* Should the time (s) remain the same ?
* Give again y or n
```

If answer is n(o) : next prompt to be expected is --> 3.8
If answer is y(es) : next prompt to be expected is --> 3.9

Else if the plotmode is H(istory) :
(see prompt 3.3)

3.14 Type of History Plots

Next the program asks for the output file and hence which plot package has to be used :

```
* To build PRESENT or OBSFIL files ?

* type 1 or P for PRESENT and
* type 2 or O for OBSFIL
```

if an error has been encountered (only p/P/o/O are allowed) in the input the program will give the message :

```
* wrong character found , try again (--> 3.14)
```

3.15 History Logfile

Next the program will ask whether the user wants to create a logfile :

```
* Give full name of extra log file
* In this file it will be written in shorthand what
```

```
* has been sent to the outputfile
*        ( blank = no extra log file )
```

Next the program checks whether the submitted SDS file is a **WAQUA** or **TRIWAQ** file and whether histories are available.

3.16 History Times Selection

First the time frame, related to the type of plot the user has been asking for, is to be determined. In case of history plots there is only one time frame possible.

The program prompts :

```
* Time frame SDS file :  File Time First    = .........
*                        File Time Interval = .........
*                        File Time Last     = .........
*
* Times should fitt within this frame
* Give return if you want to go on
*
* The following possibilities are available :
* Select by giving the indicator, and next the
* times required (all items should be separated
* from each other by a comma)
*
```

If the user specified **PRESENT** (see prompt 3.14) then two extra possibilities are available (not interesting for Histories but interesting for Verticals)

```
* A single time, give S followed by the time wanted
* Example :  S, 1230.
* More times, give M followed by the times wanted
* Example :  M, 1230. , 2460. , 2900.
```

Next possibilities are for both **PRESENT** and **OBSFIL**

```
* All times available, give A
* Example :  A
* Whole time region but with a greater time interval
* give I followed by the interval wanted
* Example :  I,550.
* Part of time region with special interval, give a
* P followed by the starting time, the interval and
* the endtime
* Example :  P,1230., 1230., 3690.
```

Whenever an error is detected, a message is printed :

```
* Start your input with one of the symbols
* S , M , I , A or P

* Try again ( --> 3.16 )
```

3.17 Output Mode Selection

For Histories there are several possibilities. The user has to decide which type of print/plot is wanted.

Remark: if the output mode selection is entered and the user has

already selected modes before, first a question whether the same modes have to be kept will be given.

```
* Do you want to keep the same data
* history plots/prints of the checkpoints
* Old choice was ........
*
* Make your choice give Y(es) or N(o)
```

So the following question is skipped if the output mode should be kept, i.e. if the users answer is Y.

```
* Which type of HISTORY print/plot ?
* Make a choice from data for :
```

Next the program shows which data are available on the selected SDS-file , i.e.

```
* Waterlevels  , give a W
* Currents     , give a C
* U-discharges , give a U
* V-discharges , give a V
* Constituent concentrations, give a P
```

If one of these types is not available one of the following messages will occur instead of one of the messages above :

```
* No Waterlevel data for Histories on SDS
* No Velocity data for Histories on SDS
* No U-discharge data for Histories on SDS
* No V-discharge data for Histories on SDS
* No Constituent Concentrations on SDS
```

Whenever an error is detected in the user input, a message is printed :

```
* Submitted character not recognized as option,
* Try again ( --> 3.17)
```

Remark : In case of **OBSFIL** a special ordering is needed. First all waterlevel plots, next all velocity plots, next all U-discharge plots, next all V-discharge plots and finally all Constituent Concentration plots. The **WAQPAN** program submits a warning when the plot asked for by the user is not allowed anymore and asks to make a new selection. So once the user has, for instance, selected currents it is not possible to return to waterlevels !

3.18 Station Selection

The user should make a selection of stations for which the selected quantities should be plotted. To this end the program gives the message :

```
* Number of stations on SDS is ...
* Make selection

* Do you want a print of all stationsnames ?
```

```
* Give Y(es) or N(o)
```

If the printmode is Y(es) this will result in a list of the form :

```
* Available stations are

* 1 = ..................... 2 = .....................
 etc.
```

3.19 Number selection

Next the numbers of the stations have to be entered :

```
* Number of stations on SDS is ...

* Give sequence numbers of stations
* that you want to select, use a comma to make
* distinction between two numbers.
*  A zero station number means that all stations have been
*  submitted
* The Stationsnumbers run from 1 to ..
* A maximum of 8 stations in one row is allowed

* Give a zero or just enter if you are ready with
* the selection of the stations
```

If a stationnumber is out of range or if no stationnumber has been specified an error message will appear and the user should try again. (--> 3.19)

3.20 Verticals / Histories Selection

Normally history plots will be produced. If it however concerns results of a **TRIWAQ** computation and if currents or constituents were specified in question given in prompt 3.17 and if a **PRESENT** file has been asked for, the user will have to answer the extra question :

```
* Which plot : Verticals or Histories ?

* Give indication what type of print/plot is wanted
* If prints/plots of verticals for each time for
* the currents, then write v or V on your screen.
* If you, however want histories then write h or H.

Of course "currents" has to be replaced by "constituents"
if appropriate.

In case that a wrong symbol to this question is submitted,
i.e. not v/V/h/H, the program gives the message :

* Error, character not recognized !
* Try again, only V or H are allowed
```

Remark : If the user selected verticals the program checks whether in the given station both currents and constituents are available. In that case all information available will be printed (included vertical viscosities, vertical diffusivities and Richardson numbers) even if the user specified only currents or only constituents. As a consequence the number of columns in the blocks may differ since in some stations

all variables are available and in other stations only part of the variables.

3.21 Depth Selection

If the user asked to make histories of either currents or constituent concentrations and the SDS-file is a **TRIWAQ** file then also the depth, at which the values have to be considered, has to be specified. Therefore the user has to enter a code to select a layer mode, i.e. a history with variable depth following the sigma-layer as specified or a history with "constant depth". When the choice "constant depth" has been made the user has to indicate a depth mode, i.e. constant depth with respect to the waterlevel or constant depth with respect to the reference level of the simulation. This information is obtained using the messages :

```
* Do you want histories along lines of variable depth
* (i.e. sigma layers ) then give s or histories along
* lines of constant depth then give d
```

If an error is detected in the layer mode, the program will continue :

```
* Only s/S/d/D allowed, try again ! ( --> 3.21)
```

Next if layer mode is s/S (sigma) :

```
* Give layer number to be considered
* Only positive integers allowed
* between 1 and <kmax>
```

and if an error is found in the layer number the program will respond :

```
  * Only positive integers allowed
  * between 1 and <kmax>

  * Try again
```

else if layer mode is d/D :

```
* Horizontal cross-sections of constant depth
* with respect to waterlevel (then give w) or
* with respect to reference level (give r)
```

If an error is encountered in the depth mode :

```
* Only w/W/r/R allowed, try again !
```

If depth mode is accepted then :

```
* Give depth to be considered
* Downwards is positive !
```

3.22 Choice for Continuation

After the creation of these History / Verticals files the program can go on to create more History plots. The user has the possibility to force the program to do so answering the following question :

```
* Do you want to continue this WAQPAN session ?
* Give y or n
```

If the answer is n(o) ---> END OF SESSION. Program strops
If the answer is y(es) ---> 3.23.

3.23 Same times Frame

The program prompts:

```
Should the time (s) remain the same ?
Give again y or n
```

If the answer is n(o): the next prompt to be expected is the one for the output time (s) (--> 3.16).
If the answer is y(es): the program replies with:

```
Same time values are used as in last loop !
```

and the next prompt to be expected is the output mode selection (--> 3.17).

Else if the plotmode is plots for MODELS-GUI :
(see prompt 3.3)

In this mode the program automatically generate the output files for *x*- and *y*-coordinates, waterdepth and waterlevels for each requested timestep (see prompt 3.25). Next to this a metafile is created. This metafile contains all kind of information which is necessary for the MODELS-GUI. The name of the metafile is determined by the run-identification of WAQPAN, whereas the extension is **def**.

3.24 Scenario name

The program will ask for a scenario name. This will be put as a title in the plots by the MODELS-GUI.

The program prompts:

```
* Give full name of scenario (max. 80 characters)
```

3.25 Map Times Selection

Next the program prompts for the selection of times (this selection is to be done by the user).

The program prompts :

```
*   Title of the SDS-file involved:
*    ....................................................
*
*
*   Time frame SDS file : File Time First    =   ........
*                         File Time Interval=   ........
*                         File Time Last     =   ........
```

```
*
*     The following possibilities are available :
*
*        a single time : time
*
*        several times (1 to N) : time1,..,timeN
*        (separate two values with a comma)
*
*        if you want all times available then give: *
*
*        if you want a new time interval then give either I (time)
*        or i (time)
*        with (time) the new interval. New multiple from ......
*        will be
*        determined by the program.
*
*     Separate times with comma's, close your input with a "/"
```

It is allowed to enter more than one line of input. The input will be closed when a slash is given on the end of a line. Per input line you can give :

- one time : this time will be added to the list of selected times
- several times : these times will be added to the list of selected times
- an asterisk : all available times will be added to the list of selected times
- interval i<dt> or I<dt> : this has the same effect as giving an asterisk, only the new given time interval dt will be used. Note that no spaces in between **i** and **dt** are allowed

Every time that an input line is given, the program shows the selected time(s) i.e. one or more times the following line if you chose one of the first two possibilities :

```
Added <time>
```

and the lines :

```
times added with interval <time interval>
starting with <time first>
ending with <time last>
```

if you chose the third possibility.

In the program the next checks are done and (if needed) adaptions are made during the analysis of your input :

- A single time must fit within the time range. When it does, the nearest time available is added.

- A new time interval is rounded to a whole number of time steps.

Whenever an error is detected, a message is printed and the rest of the input line is analyzed further.

3.26 Code Selection

The user has to enter a (or several) code (s) to select the type of data to be outputted. All input must be given on one line, with the choices separated by comma's only. An input error results in a message after which the complete line must be entered again. Depending on which choice the user made and what kind of model has been used one of the following menu's will show up.

```
26A      WAQUA

    * E         = Magnitude of local Velocity
    * U         = Physical U-velocity component in wl point
    * V         = Physical V-velocity component in wl point
    * G<nr>     = Concentration of Constituent number <nr>


26B      TRIWAQ

    * B         = Bottom Friction
    * E         = Magnitude of local Velocity
    * U         = Physical U-velocity component in wl point
    * V         = Physical V-velocity component in wl point
    * F         = Eddy Viscosity
    * G<nr>     = Concentration of Constituent number <nr>
    * H         = Eddy Diffusivity
    * I         = Richardson numbers

26C      WAQAD

    * P         = Adjoint waterlevels
    * Q         = Adjoint U-velocity component in wl point
    * R         = Adjoint V-velocity component in wl point
```

3.27 Choice to Continue

After the creation of these map files the program can go on to create more map plots. The user has the possibility to force the program to do so answering the following question :

```
 * Do you want to continue this WAQPAN session ?
 * Give y or n
```

If answer is n(o) ----> END OF SESSION. Program stops.
If answer is y(es) ----> 3.28

3.28 Same times Frame ?

If the answer for continuation was yes then the program starts again but first it is checked whether a time selection has to be done again or not :

```
 * Should the time (s) remain the same ?
 * Give again y or n
```

If answer is n(o) : next prompt to be expected is --> 3.25
If answer is y(es) : next prompt to be expected is --> 3.26

4 Technical Description Routines WAQPAN

The program **WAQPAN** offers a **SIMONA** user the opportunity to read information from the **SDS**-file, make combinations of the various quantities and create (new) files in such a way that they can be used as input files for one of the plotpackages **PRESENT** , **ANIMATE**, **SIMVIEW**, **MATLAB/MODELS-GUI** or **HISPLO** (the latter via the interface **OBSFIL**).
In this chapter a short explanation is given, especially for the maintenance of **WAQPAN**, which quantities are treated in which subroutines of **WAQPAN**. Also a short description of the task of each subroutine is given. This description is the same as given in the header of each routine following the key **DESCRIPTION**.

In routine **WANP02** the user is asked which type of datafile has to be created : (**lanima** is the variable to indicate the various possibilities)

lanima = 1 : **PRESENT - MAP**
2 : **ANIMATE - MAP**
3 : **PRESENT - HISTORIES**
4 : **OBSFIL - HISTORIES**
8 : **BOX - MAP**
9 : **MATLAB - MAP**

In the MAP part, i.e. plots of fielddata for a given time, there are several possibilities. First the user should indicate whether he is interested in :

- actual, i.e. momentary quantities like for instance waterlevels, or
- integrated, i.e. residual quantities like Eulerian or Langrangian transports, or
- tidal constants, i.e. mean levels, astronomical amplitudes and local phase lags, or
- adjoint variables, i.e. adjoint waterlevels and adjoint velocity components

Although, the tidal constants are time-independent, they are treated as fielddata.

For actual map-quantities (fielddata), except the tidal constants and adjoint variables, and the **TRIWAQ** mode the user may choose between horizontal or vertical cross-sections.
In the case of HISTORIES and lanima = 3 there is the possibility to make plots "over the vertical" of either velocities or constituent concentrations for a selected location (station).

The MAPDATA times are read, both for actual and for residual quantities in routine **WANP14**. Times for HISTORYDATA are read in routine **WANP33**. The geometry window that will be used in the case of MAPDATA is determined in routine **WANP28** and the determination of stationnames in the case of HISTORYDATA takes place in routine **WANP31**.

Code selection is performed in routine **WANP13**.

The outputfiles are being filled in routine **WANP06** for the MAP-plots and in routine **WANP32** for the HISTORY-plots.

After the execution of all these tasks the user may change (if wanted) times, windows, stations and codes, and either add new blocks to the outputfiles (in the case of **PRESENT** and **HISPLO**) or create new sub-files (in the case of **ANIMATE, BOX, MATLAB** or **MODELS-GUI**).

In the case of Eulerian residual quantities the following options are available :

1. Residual velocity and/or transport in a 3D sigma-layer (**WANP51**).
2. The (interpolated) residual velocity- or transport in case of a 3D situation but with a constant depth or in case of a 2D situation (**WANP57**, **WANP58**).

In the case lanima= 2 and Eulerian quantities the user has the options:

1. Residual velocity and/or transport in a 3D sigma-layer (**WANP52**).
2. The (interpolated) residual velocity or transport in a 3D situation with constant depth or in a 2D computation (**WANP56**, **WANP57**).

In the case lanima = 1 and Lagrangian there is only one possibility :

1. Lagrangian (residual)-displacements in a 3D sigma-layer (**WANP53**).

If lanima = 2 and the option is Lagrangian there is also only one, the same, possibility :

1. (residual)-displacements in a 3D sigma-layer (**WANP54**).

In the case of actual (not integrated over time and/or avaraged) quantities including the tidal constants and adjoint variables the following possibilities are available for lanima = 1 (**PRESENT**) :

- In case of a horizontal cross-section of a 3D domain (**WANP19**) with code :

SE - Coordinates waterlevel points, waterlevels, bottomfriction and Chezy values
H - Coordinates depth points and the depth in those points
VC - Coordinates waterlevel points,U-, V-, Omega- and W-velocities, vertical viscosity, vertical diffusivity, Richardson numbers and concentrations of lmax constituents
TC - Mean waterlevels or currents of tide, astronomical, amplitudes and local phase lags
AH - Adjoint waterlevels and physical U- and V-velocities

- If it concerns results of a 2D computation or a horizontal cross-section of a 3D model for a constant depth (routine **WANP21**):

SE - Coordinates waterlevel points and the waterlevel
H - Coordinates depth points and the local depth values
VC - Coordinates waterlevel points,U- and V-velocities and concentrations of lmax constituents
TC - Mean waterlevels or currents of tide, astronomical, amplitudes and local phase lags
AH - Adjoint waterlevels and physical U- and V-velocities

- If it concerns a vertical cross-section (**WANP20**) with code :

SD - The local coordinate (in this case a line), the local depth, the waterlevel, the local discharge, the local bottomfriction-velocity and the local Chezy value. All quantities computed in waterlevel points.
VC - The local horizontal and vertical coordinate (in this case it concerns a plane), the layer depth for each point, the local, i.e. for that layer, U-, V-, Omega- and W-velocity and for each point from the cross-section the vertical viscosity, diffusivity and Richardson numbers. Finally, if available, the concentrations of lmax constituents.

If lanima = 2 or 9, i.e. **ANIMATE** or **MATLAB,** the following possibilities are offered to the user :

- If it concerns a horizontal cross-section in a 3D model then (see routine **WANP08**) the codes are :

A[V] The waterlevel
B[V] The bottomfriction-velocity in waterlevel points
C[V] The depth values in waterlevel points
E[V] The velocity magnitude in waterlevel points
F[V] The eddy-viscosity in the waterlevel points
G[V] The concentration of the constituent considered in the waterlevel points
H[V] The eddy-diffusivity in the waterlevel points
I[V] The Richardson numbers in the waterlevel points
M Mean waterlevels or currents of tide in the waterlevel points
N Astronomical amplitudes in the waterlevel points
O Local phase lags in the waterlevel points
P The adjoint waterlevel
Q The physical adjoint U-velocity component in waterlevel point
R The physical adjoint V-velocity component in waterlevel point

- If it concerns a 2D model, only the options A[V], C[V], E[V], G[V], M, N, O, P, Q and R mentioned in the above list are available (see routine **WANP10**).

- If it concerns a vertical cross-section (3D, see routine **WANP09**) only the options E[V], F[V], G[V], H[V] and I[V] mentioned in the above list are available. The corresponding local coordinates (CCO) and boundary outlines (LIN) files are constructed.

In the above list, a [V] means that the velocity-vector plot may be added to the mapplots.

The coordinatesfile CCO and boundary outlines-file LIN are created separately (see routines **WANP29** and **WANP30**).

In routine **WANP67** a list of tidal components with their names is created and the user should make a selection of these components.

In routine **WANP72** a metafile with extension **def** meant for the **MODELS-GUI** is created.

In the case lanima = 3 (**PRESENT**) or lanima = 4 (**HISPLO**) the following possibilities are available in **WAQPAN**. We have the history plots of:

W - Waterlevels
C - Currents
U - U-discharges
V - V-discharges
P - Concentrations

The user should also indicate the station-numbers (names). This is done in routine **WANP31**. If currents or concentrations are asked for in 3D, these can also be considered over the vertical. Therefore the user should also indicate whether (only if lanima = 3) HISTORIES or VERTICALS are required (see routine **WANP39**). In the case of HISTORIES and either currents (C) or concentrations (P), the user should indicate whether the values have to be computed per sigma layer or for a constant depth (see routine **WANP04**).

In routine **WANP32**, SDS results are elaborated for the HISTORIES files. In routine **WANP34**, results are transported into helparray DATAR and printed as a **PRESENT** file (if lanima = 3, see routine **WANP35**) or printed as a **HISPLO** file (if lanima = 4, see routine **WANP36**).

In routine **WANP40** the elaboration of results, if verticals have been asked for, is performed.

The routines named so far play a key-role in the **WAQPAN** package. All other routines are helproutines that are called only once. All routines are easy to understand.

Finally a list of all routines with their **DESCRIPTION** is given.

Main program : WAQPAN

Program to read 2D **WAQUA** or 3D **TRIWAQ** simulation data from

the SIMONA DATA STORAGE (SDS) file and to prepare and generate (an) INPUT file(s) for the packages **PRESENT** (Map or Time), **ANIMATE** (Map), **HISPLO** (Time histories), **MATLAB** (Map) or **MODELS-GUI** (Map).

Subsequently the following tasks are performed :

- Check whether the user wants to create a file suited for use by **PRESENT** , **HISPLO**, **ANIMATE**, **BOX** or **MATLAB**
- Ask user to specify the name of the SDS-file and of the experiment name. Open file and experiment
- Place all important time-independent arrays in core
- Ask user to specify the time steps for which output is wanted
- If Map then

    Check whether the user wants to create a file suited for **PRESENT**, **ANIMATE or MATLAB**

    Ask user what type of cross-section is wanted

  Else if History then

    Check whether the user wants to create a file suited for **PRESENT** or **HISPLO**

    Ask user which stations have to be considered

  Endif

- Ask user which codes (i.e. quantities) are required
- Execute a loop for all times. Determine for each time the values for the codes as specified in the window (MAP) or in the stations (HISTORY) as specified
- Ask whether user wants to specify and run other times, windows and/or codes

Description of all subroutines of WAQPAN

Subroutine WANFIN

Initialisation of computational arrays for FLOW part

Subroutine WANLAV

Computes the position of the layer-interfaces at velocity points

Subroutine WANP01

Determine name SDS file and experiment. Check whether file and ex-

periment exist.

Subroutine WANP02

Determine which print/plot is asked for by user. If needed open PRESENT file.

Subroutine WANP03

Read all time independent arrays from SDS and place pointers in array starting from indicated positions.

Subroutine WANP04

Determine type of horizontal cross- section: along sigma interface or along plane with constant depth to reference plane.

Subroutine WANP05

Write explanation of **ANIMATE** file names to log file.

Subroutine WANP06

Run through all times and all wanted codes and make for the geometry wanted the corresponding **ANIMATE** or **PRESENT** files.

Subroutine WANP07

Actualize waterlevel flag with the value 99999. on dry points and 0. elsewhere.

Subroutine WANP08

Write contents of arrays in the correct format to a file to be used by program **ANIMATE**.

Subroutine WANP09

Create and fill files with data to be used by the **ANIMATE** program in the case of vertical cross-sections.

Subroutine WANP10

Write contents of arrays either of **WAQUA** or of a cross-section of constant depth of **TRIWAQ** in the correct format to a file to be used by program **ANIMATE**.

Subroutine WANP11

Compute all derived quantities from the main arrays from **WAQUA** or **TRIWAQ** as read from SDS. These arrays can be used to print and/or plot results of computati-ons by for instance **PRESENT** or **ANIMATE**.

Subroutine WANP12

Explode one-dimensional arrays into full twodimensional arrays.

Subroutine WANP13

Get code info from standard input for MAP data.

Subroutine WANP14

Read times from standard input that fitt within the given range and time increment.

Subroutine WANP15

Determine the global windfield for the time CURTIM.

Subroutine WANP16

Initialize all computational points with 0 and all dry points with 9999.

Subroutine WANP17

Determine julian day number of given date. It is assumed that a realistic date is given.

Subroutine WANP18

Compute date and time corresponding to integration step nst in the form yy/mm/ddhh:mm:ss.

Subroutine WANP19

Print results in case of a horizontal cross-section to a file to be used by **PRESENT**.

Subroutine WANP20

Print data on a file in such a way that this file can be used to make plots/prints by **PRESENT**.

Subroutine WANP21

Print results in case of **WAQUA** or a hori-zontal layer with constant depth to a file to be used for **PRESENT**.

Subroutine WANP22

Helproutine to compute the values of concentrations, u-velocities and v-velocities in a 2D plane of constant depth. The depth is measured either below reference plane or below the current waterlevel from the values as given in a general 3D field.

Subroutine WANP23

Determine local min and max values for looking window.

Subroutine WANP24

Copy one value (realin) to another value (reaout).

Subroutine WANP25

Compute helparrays for a curvilinear transformation.

Subroutine WANP26

Update all values with respect to waterlevel and depth.

Subroutine WANP27

Initialize arrays to be used in the generation of vertical cross-sections.

Subroutine WANP28

Determine the window coordinates from standard input.

Subroutine WANP29

Write curvilinear grid coordinates (of the **WAQUA** depth positions) to the output CCO (Curvilinear Coordinates Output) file.

Subroutine WANP30

Write grid position of boundary outlines to output file.

Subroutine WANP31

Read and select data (Waterlevel, Current, Discharges or Transport) together with the Checkpoints, i.e. station numbers from standard input. Make difference between **PRESENT** and **OBSFIL**.

Subroutine WANP32

Run through all times for all stations and make printfiles for either **PRESENT** or **OBSFIL**. Print according to the submitted code.

Subroutine WANP33

Read times from standard input that fitt within the range and time increment given. Place all times in array.

Subroutine WANP34

Run through all times for all stations and fill array for either **PRESENT** or **OBSFIL**.

Subroutine WANP35

Run through all times for all stations and make printfiles for **PRESENT**. Print according to the submitted code.

Subroutine WANP36

Run through all times for all stations and make printfiles for **OBSFIL**. Print according to the submitted code.

Subroutine WANP37

Determine the angle with the north direction in degrees for the vector (ucompo, vcompo).

Subroutine WANP38

Copy temporary **OBSFIL** file to permanent **OBSFIL** file.

Subroutine WANP39

Determine wether user wants plots of verticals for each time and each station or that user wants plots of histories for all times for all stations.

Subroutine WANP40

Run through all times for all stations and make printfiles for the verticals to be used by **PRESENT**. Print according to the submitted code.

Subroutine WANP41

Determine important variables like vicow, difcw, rich and rho for specified location and time.

Subroutine WANP42

Write results as asked by user to **PRESENT** file.

Subroutine WANP43

Compute, dependent on the way the user has defined the bottom roughness the related Chezy3D coefficient.

Subroutine WANP44

Compute densities after salinity- or temperature computation.

Subroutine WANP45

Compute bottom stress velocity and viscosity coefficients using k-eps model.

Subroutine WANP46

Compute diffusivity for all layers.

Subroutine WANP47

Let user select station number.

Subroutine WANP51

Print Eulerian residual results in case of a horizontal sigma cross-section to a file to be used by **PRESENT**.

Subroutine WANP52

Write contents of Eulerian residual velocity and transport arrays in the correct format to a file to be used by program **ANIMATE**.

Subroutine WANP53

Write contents of Lagrangian residual displacement arrays in the correct format to a file to be used by program **ANIMATE**.

Subroutine WANP54

Print Lagrangian residual results in case of a horizontal sigma cross-section to a file to be used by **PRESENT**.

Subroutine WANP55

Ask user which MAP is wanted in case of Eulerian residuals for program **ANIMATE**.

Subroutine WANP56

Write contents of Eulerian residual velocity or Transport arrays in 2D in the correct format to a file to be used by program **ANIMATE**.

Subroutine WANP57

Compute the values of Eulerian residual velocities, and transports in a 2D plane of a constant depth from values given in a general 3D field. The depth is measured either below reference plane or below the waterlevel.

Subroutine WANP58

Print Eulerian residual results in case of **WAQUA** or horizontal layer with constant depth to a file to be used by **PRESENT**.

Subroutine WANP59

Checks if meshes of two different experiments are the same by

comparing NMAX, MMAX and LGRID.

Subroutine WANP60

Gets MAP times from CONTROL arrays.

Subroutine WANP61

Computes the difference fields of arrays SEP, UP, VP and RP.

Subroutine WANP62

Defines the block code to be printed in PRESENT.OUT.

Subroutine WANP63

Writes the bulk data arrays to the Animate, Box or Matlab files.

Subroutine WANP64

Writes the arrays KHU and KHV to the Matlab files for plotting drying/flooding.

Subroutine WANP65

Compute the differences of waterlevel and constituents from two SDS-files or two layers

Subroutine WANP66

Determine type of difference map (absolute, nominal or fractional)

Subroutine WANP67

Reads and selects tidal components from standard input.

Subroutine WANP68

Write the data arrays to a binary MAT-file

Subroutine WANP69

Fills the array homeg with angular velocities and their component numbers

In case of astronomical splitting fill extra angular velocities corresponding to tidal components P1 (33), NU2 (60), K2 (79) and LABDA2 (70).

Subroutine WANP70

Fills an 1D lgrid-array equivalent of passed full matrix array in case of adjoint variables

Subroutine WANP71

Write grid coordinates (of the WAQUA depth positions) to the binary MAT-files meant for GUI

Subroutine WANP72

Write relevant quantities to definition file

Description of non-WAQPAN routines

Subroutine TRSCHO

Calculation of the actual Chezy-values depending a resistance formula or the log-law of the wall.

Subroutine TRSVIO

In this routine an algebraic turbulence model is implemented for the calculation of the vertical eddy-viscosity coefficient.

Subroutine USTAR

Computes the ratio s = u/ustar with ustar the friction velocity based on the physical horizontal velocity u at distance zk to the wall with logarithmic or linear law of the wall.
It also takes into account of the wall roughness through the roughness parameter z0

Subroutine WAGI2B

Calculates 32-bit representation of an integer number

Subroutine WAGR2B

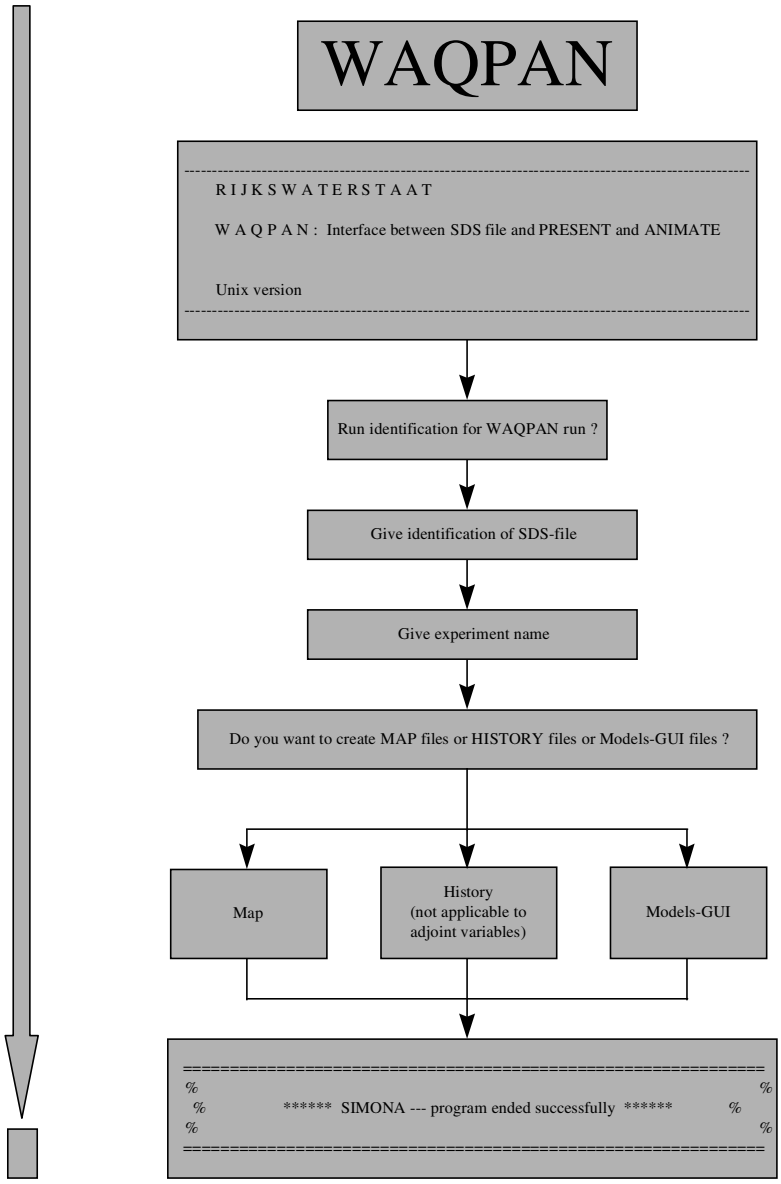
Calculates 32-bit representation of a floating-point number

Appendix A Example of metafile for Models-GUI

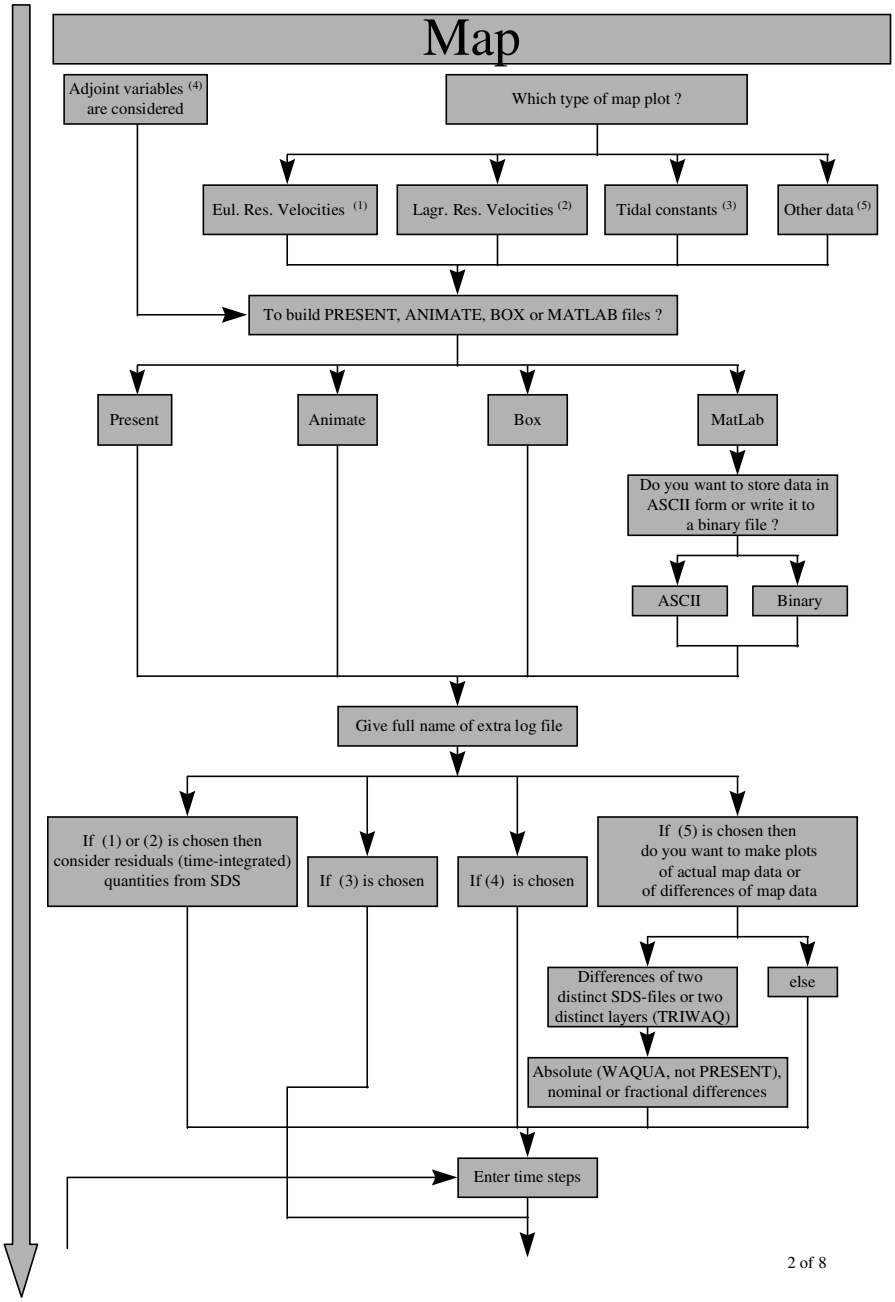
```
% G98.DEF
% Invoer bestand met instellingen voor de Modellen-GUI
% Dit bestand is aangemaakt met WAQPAN

% RIKZ / Rijkswaterstaat 1998

% Korte omschrijving van het model
%     RUN-ID   = GREV98  , OPC = T00

% Instellingen voor de Modellen-GUI

% Versie 1.0, december 1998
% Matlab 5.2

% scenario naam afgekort, zoals deze in de datafilenaam voorkomt
param_struct.scen = 'g98';

% scenario naam voluit, bv om boven plot te plaatsen
param_struct.scen_naam = 'Grevelingen 1998';

% variabele voor het samenstellen van de filenamen
param_struct.file_format = [ 3 2 2 2];

% betekenis van constituenten
param_struct.G1 = 'salinity';

% aantal lagen van het model
param_struct.aant_laag = 5;

% verdeling van diepte in sigma en vaste lagen

% sigma_laag in fracties van het geheel -> som(sigma) == 1
param_struct.sigma_laag = [ 0.0000 60.0000 30.0000 .0000 …
10.0000]/100;

% vaste_laag in [m]
param_struct.vaste_laag = [    5.0000      .0000      .0000     2.0000 …
.0000];

% datum en tijd variabelen:

% begin_datum van het experiment
param_struct.begin_datum = ' 1 DEC 1998     ';

% begin_tijd van de weggeschreven resultaten van het experiment
% in [minuten] tov begin_datum 00:00
param_struct.begin_tijd  =    720;

% tijdcycles zoals ze in de filenamen voorkomen
param_struct.tijd_file = [   1    2    3    4    5    6    7 …
8    9    10];

% tijd in werkelijkheid in [minuten]
% tov begin_datum + begin_tijd
```

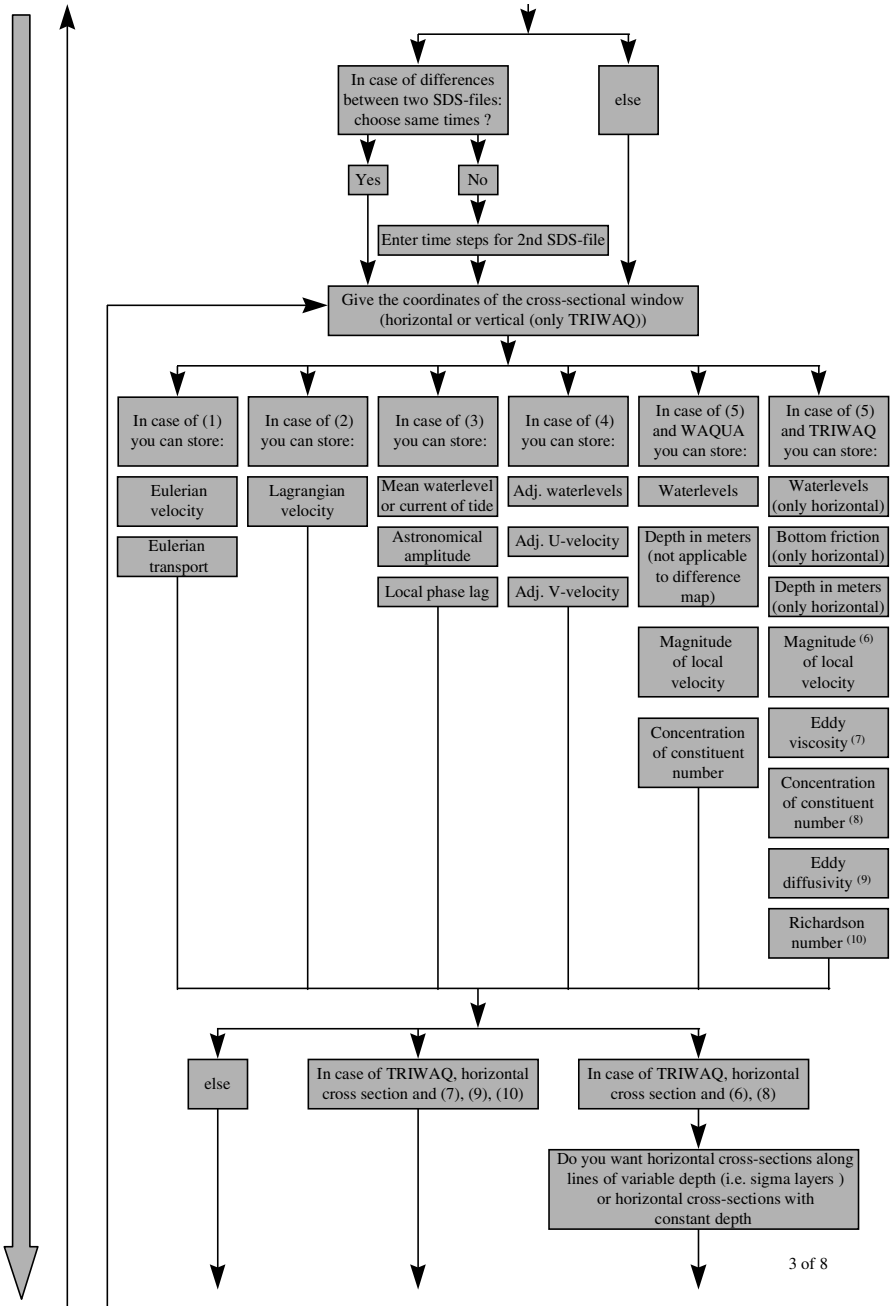
```
param_struct.tijd_real = [    0     30     60     90     120     150 …
180    210    240    270];
```

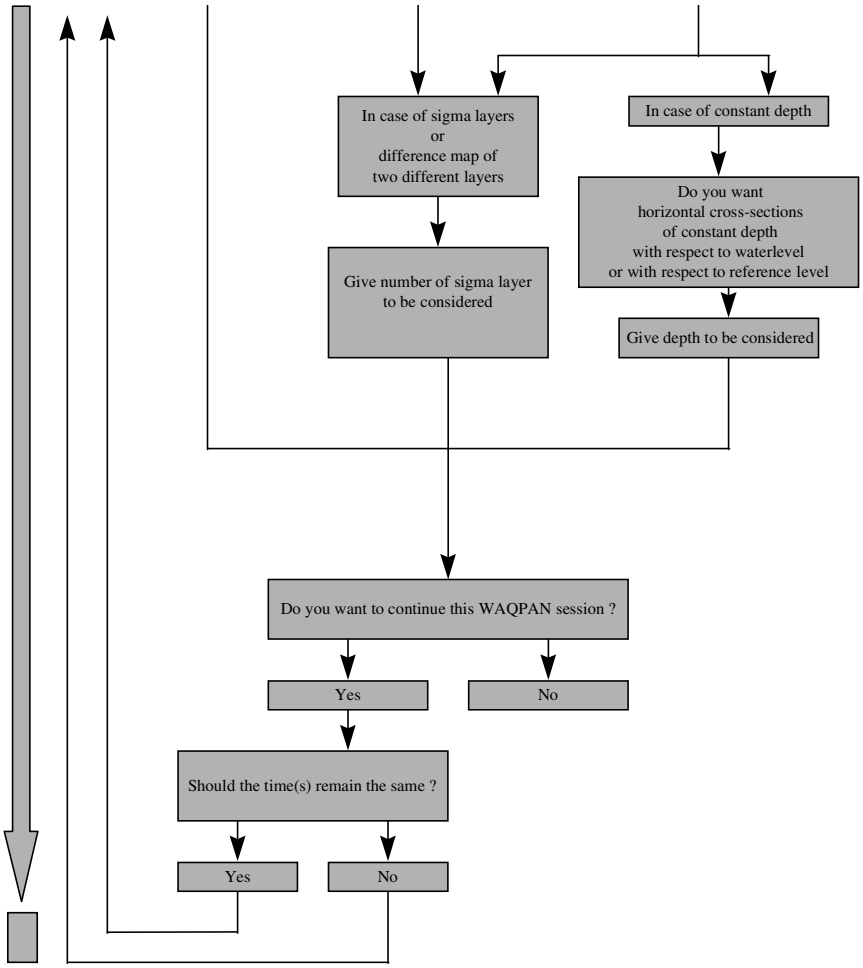
Appendix B Flow diagram



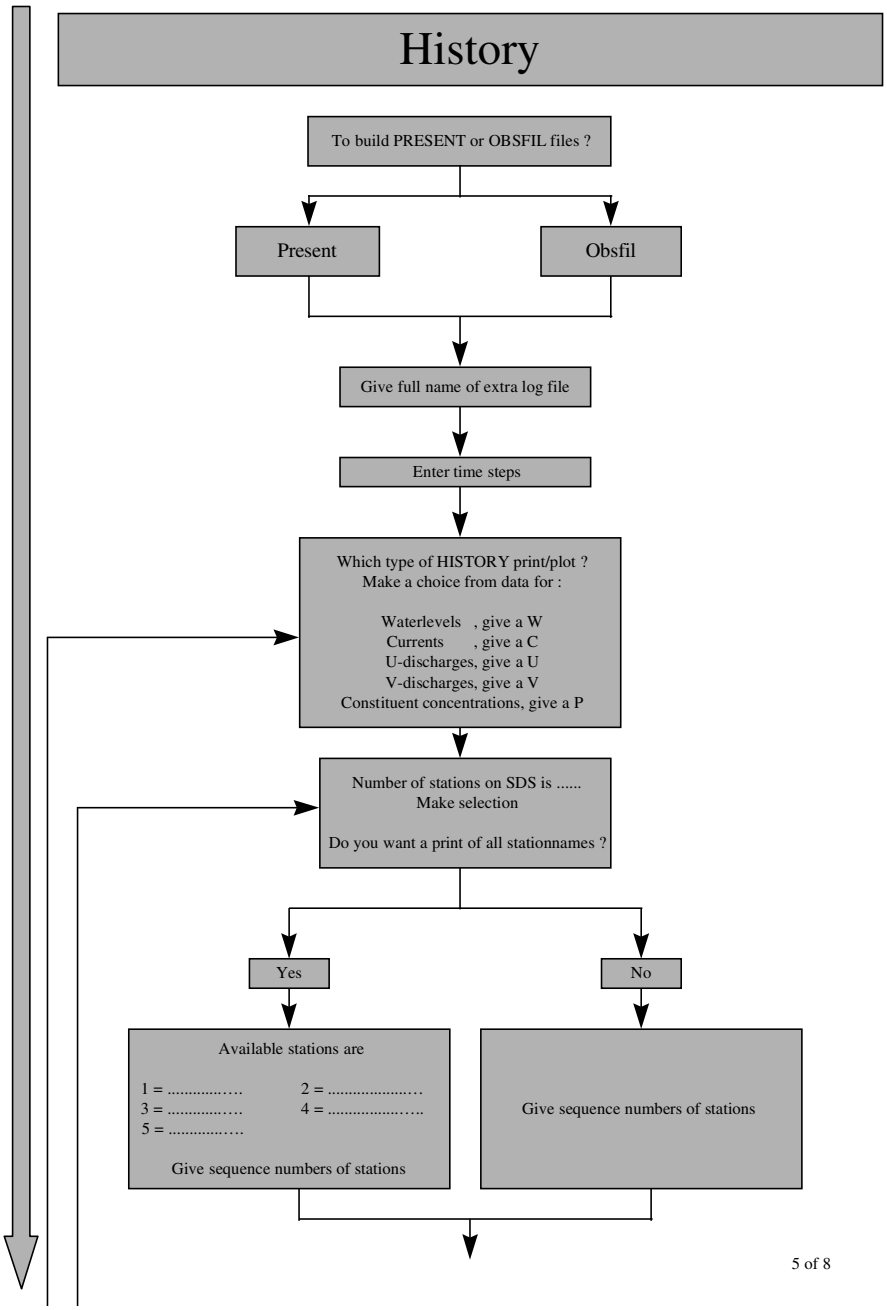
1 of 8

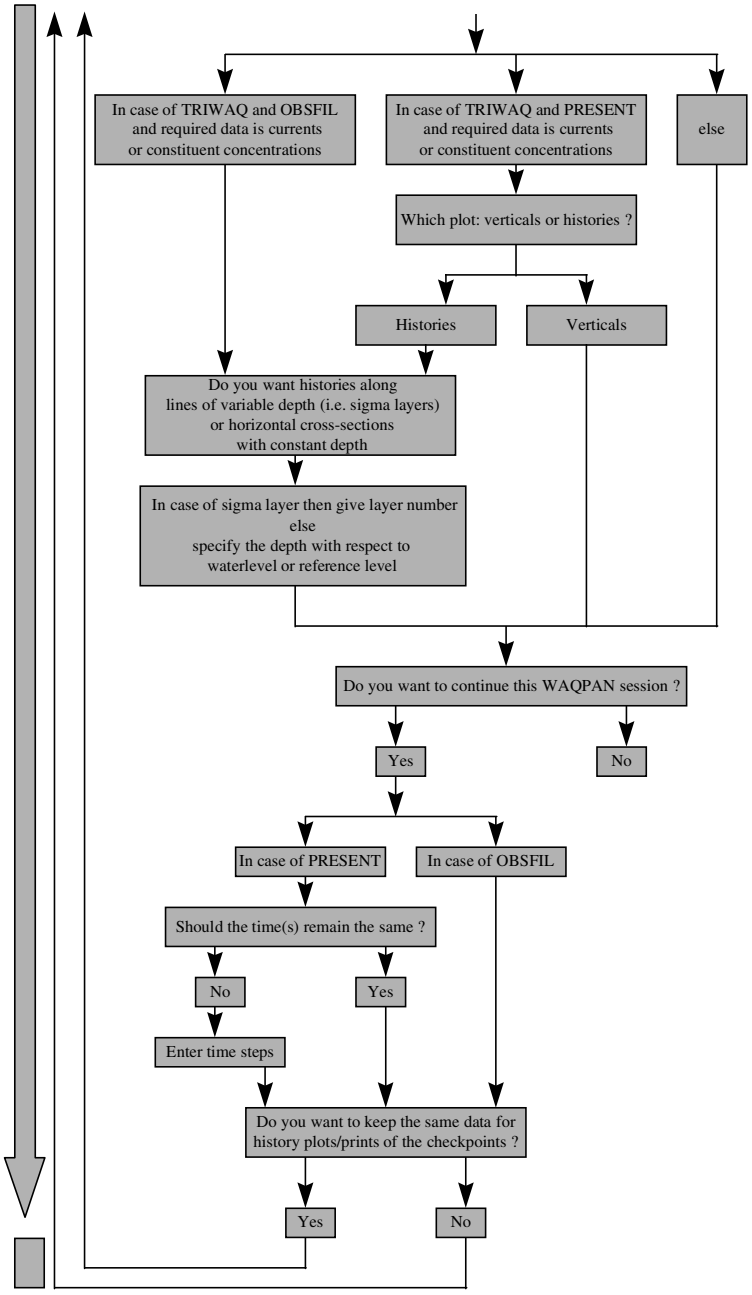

Map
Adjoint variables (4) are considered
Which type of map plot ?
Eul. Res. Velocities (1)
Lagr. Res. Velocities (2)
Tidal constants (3)
Other data (5)
To build PRESENT, ANIMATE, BOX or MATLAB files ?
Present
Animate
Box
MatLab
Do you want to store data in ASCII form or write it to a binary file ?
ASCII
Binary
Give full name of extra log file
If (1) or (2) is chosen then consider residuals (time-integrated) quantities from SDS
If (3) is chosen
If (4) is chosen
If (5) is chosen then do you want to make plots of actual map data or of differences of map data
Differences of two distinct SDS-files or two distinct layers (TRIWAQ)
else
Absolute (WAQUA, not PRESENT), nominal or fractional differences
Enter time steps
2 of 8


In case of differences between two SDS-files: choose same times ?
else
Yes
No
Enter time steps for 2nd SDS-file
Give the coordinates of the cross-sectional window (horizontal or vertical (only TRIWAQ))
In case of (1) you can store:
Eulerian velocity
Eulerian transport
In case of (2) you can store:
Lagrangian velocity
In case of (3) you can store:
Mean waterlevel or current of tide
Astronomical amplitude
Local phase lag
In case of (4) you can store:
Adj. waterlevels
Adj. U-velocity
Adj. V-velocity
In case of (5) and WAQUA you can store:
Waterlevels
Depth in meters (not applicable to difference map)
Magnitude of local velocity
Concentration of constituent number
In case of (5) and TRIWAQ you can store:
Waterlevels (only horizontal)
Bottom friction (only horizontal)
Depth in meters (only horizontal)
Magnitude (6) of local velocity
Eddy viscosity (7)
Concentration of constituent number (8)
Eddy diffusivity (9)
Richardson number (10)
else
In case of TRIWAQ, horizontal cross section and (7), (9), (10)
In case of TRIWAQ, horizontal cross section and (6), (8)
Do you want horizontal cross-sections along lines of variable depth (i.e. sigma layers) or horizontal cross-sections with constant depth
3 of 8

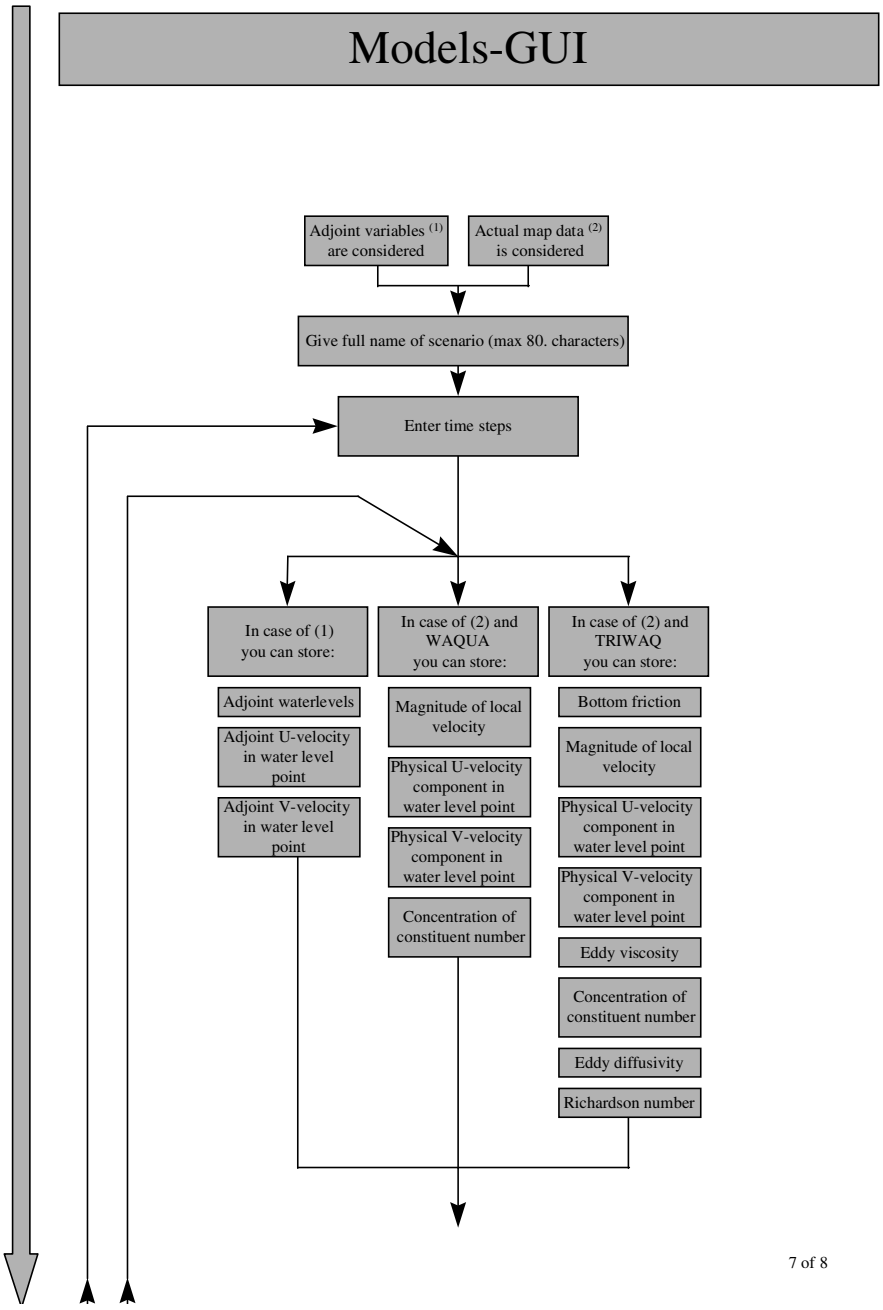

In case of sigma layers
or
difference map of
two different layers
In case of constant depth
Do you want
horizontal cross-sections
of constant depth
with respect to waterlevel
or with respect to reference level
Give number of sigma layer
to be considered
Give depth to be considered
Do you want to continue this WAQPAN session ?
Yes
No
Should the time(s) remain the same ?
Yes
No

4 of 8

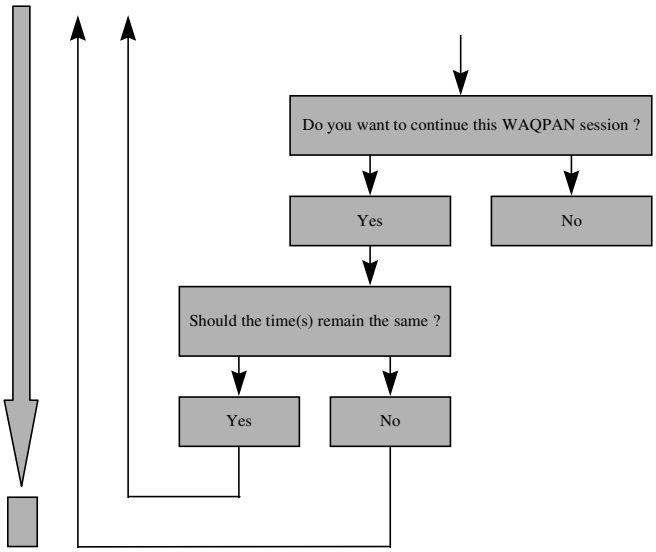

History
To build PRESENT or OBSFIL files ?
Present
Obsfil
Give full name of extra log file
Enter time steps
Which type of HISTORY print/plot ?
Make a choice from data for :
Waterlevels , give a W
Currents , give a C
U-discharges, give a U
V-discharges, give a V
Constituent concentrations, give a P
Number of stations on SDS is .....
Make selection
Do you want a print of all stationnames ?
Yes
No
Available stations are
1 = ...
2 = ...
3 = ...
4 = ...
5 = ...
Give sequence numbers of stations
Give sequence numbers of stations
5 of 8


In case of TRIWAQ and OBSFIL
and required data is currents
or constituent concentrations
In case of TRIWAQ and PRESENT
and required data is currents
or constituent concentrations
else
Which plot: verticals or histories ?
Histories
Verticals
Do you want histories along
lines of variable depth (i.e. sigma layers)
or horizontal cross-sections
with constant depth
In case of sigma layer then give layer number
else
specify the depth with respect to
waterlevel or reference level
Do you want to continue this WAQPAN session ?
Yes
No
In case of PRESENT
In case of OBSFIL
Should the time(s) remain the same ?
No
Yes
Enter time steps
Do you want to keep the same data for
history plots/prints of the checkpoints ?
Yes
No

6 of 8


Models-GUI
Adjoint variables (1) are considered
Actual map data (2) is considered
Give full name of scenario (max 80. characters)
Enter time steps
In case of (1) you can store:
In case of (2) and WAQUA you can store:
In case of (2) and TRIWAQ you can store:
Adjoint waterlevels
Adjoint U-velocity in water level point
Adjoint V-velocity in water level point
Magnitude of local velocity
Physical U-velocity component in water level point
Physical V-velocity component in water level point
Concentration of constituent number
Bottom friction
Magnitude of local velocity
Physical U-velocity component in water level point
Physical V-velocity component in water level point
Eddy viscosity
Concentration of constituent number
Eddy diffusivity
Richardson number

7 of 8


Do you want to continue this WAQPAN session ?
Yes
No
Should the time(s) remain the same ?
Yes
No

8 of 8

Glossary

ANIMATE Presentation program for personal computers with MS-Windows that enables you to visualize calculations from mathematical models using animations.

BOX Output option of which the result can be used in SIMONA-input files.

MATLAB MATLAB is an interactive program to help you with numeric computation and data visualisation. Principally, MATLAB is built upon a foundation of sophisticated matrix software for analysing linear systems of equations.

HISPLO Presentation program to plot time histories from at most twenty different Dadi files. Subprogram of the WAQUA package. (WAQUA43)

OBSFIL Program that stores observed data that have a regular time interval in a file, designed for selective retrieval and display by the HISPLO system. Subprogram of the WAQUA package. (WAQUA32)

PRESENT Presentation program developed by Delft Hydraulics and available under UNIX and on personal computers that enables you to visualize results of calculations from mathematical models. The program needs three files in order to make plots :

1) a data file (for example PRESENT.OUT)
2) an INPUT file in which it has been written which tasks have to be performed
3) a configuration file

MODELS-GUI Graphics User Interface for MATLAB (version 5.2) developed by Xi Visualisatie and available under Windows-95 and UNIX. This program enables the user to visualize results of WAQUA/TRIWAQ/WAQAD. This program makes horizontal cross-section at arbitrary depth or vertical cross-sections. Also an animation of some data is possible.

Index